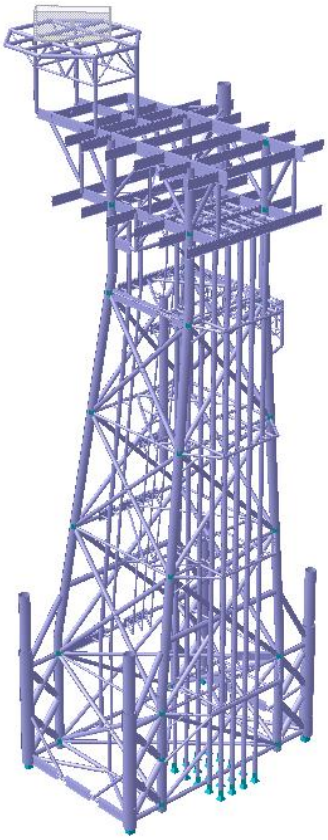
SESAM™
USER MANUAL

GENIE VOL. IV
CODE CHECKING OF BEAMS



Concept design and analysis
of offshore structures

DET NORSKE VERITAS

This page is blank for printing

Sesam™
User Manual
GeniE Vol. IV

Code checking of beams

Concept design and analysis
of offshore structures

15 September 2011

Valid from program version 6.0

Developed and Marketed by
DET NORSKE VERITAS

DNV Software Report No.: 00-000 / Revision 0, 15 September 2011

Copyright © 2011 Det Norske Veritas Software

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher.

Published by:

Det Norske Veritas Software
Veritasveien 1
N-1322 HØVIK
Norway

Telephone: +47 67 57 99 00

Facsimile: +47 67 57 72 72

E-mail, sales: software.sesam@dnv.com

E-mail, support: software.support@dnv.com

Website: www.dnvsoftware.com

If any person suffers loss or damage which is proved to have been caused by any negligent act or omission of Det Norske Veritas, then Det Norske Veritas shall pay compensation to such person for his proved direct loss or damage. However, the compensation shall not exceed an amount equal to ten times the fee charged for the service in question, provided that the maximum compensation shall never exceed USD 2 millions. In this provision "Det Norske Veritas" shall mean the Foundation Det Norske Veritas as well as all its subsidiaries, directors, officers, employees, agents and any other acting on behalf of Det Norske Veritas.

GeniE User Manual
Code checking of beams
Version 6.0

Table of Contents

1. INTRODUCTION – CODE CHECKING OF BEAMS 5

1.1 HOW TO READ THIS MANUAL 5
1.2 LEARNING FROM TUTORIALS FOR CODE CHECKING 5
1.3 ACRONYMS FREQUENTLY USED IN THE USER MANUAL 7
1.4 CODE CHECKING ON LARGE MODELS 8

2. BEAM CODE CHECKING FEATURES OF GENIE 9

2.1 CODE CHECKING STANDARDS 9
2.2 AVAILABLE CODE CHECKS 10
2.3 MODEL REQUIREMENTS 12
2.4 CODE CHECKING PROCEDURE 13
2.4.1 A reference model 14
2.4.2 Making load combinations 14
2.4.3 Create a capacity manager 15
2.4.4 Create capacity members and joints 15
2.4.5 Create code check runs 16
2.4.6 Compute code checking forces 17
2.4.7 Perform the code check 18
2.4.8 Making a code checking report 19

3. HOW TO DO BEAM CODE CHECKING 21

3.1 CREATE A CAPACITY MANAGER 21
3.2 DEFINE MEMBER AND JOINTS 22
3.3 CREATE A CODE CHECK RUN 25
3.3.1 Define global general code checking parameters 27
3.3.1.1 General parameters API WSD 2002&2005 28
3.3.1.2 General parameters API LRFD 2003 29
3.3.1.3 General parameters NORSOK N004 30
3.3.1.4 General parameters ISO 19902 2007 30
3.3.1.5 General parameters AISC ASD 2005 31
3.3.1.6 General parameters AISC LRFD 2005 32
3.3.1.7 General parameters EUROCODE 3 EN 1993-1-1 32
3.3.1.8 General parameters DANISH STANDARD 412 / 449 33
3.3.2 Define global member parameters 35
3.3.2.1 Member parameters API WSD 2002 & 2005 36
3.3.2.2 Member parameters API LRFD 2003 37
3.3.2.3 Member parameters NORSOK N004 38
3.3.2.4 Member parameters ISO 19902 2007 39
3.3.2.5 Member parameters AISC ASD 2005 40
3.3.2.6 Member parameters AISC LRFD 2005 42
3.3.2.7 Member parameters EUROCODE 3 EN 1993-1-1 43
3.3.2.8 Member parameters DANISH STANDARD 412 / 449 45
3.3.3 Define global joint parameters 47
3.3.3.1 Brace type classification 50
3.3.3.2 Gap classification 51
3.3.3.3 Load transfer classification 51
3.3.3.4 Through brace classification 51
3.3.3.5 Weld thickness classification 51
3.3.3.6 Brace utilisation 51

3.4 LOCAL CODE CHECKING PARAMETERS 52
3.4.1 Modify local member parameters *52*
3.4.2 Modify local joint parameters *52*
3.5 CODE CHECKING POSITIONS AND FORCES 54
3.6 PERFORM THE CODE CHECK 56
3.7 INVESTIGATE THE RESULTS 57
3.7.1 From the browser *57*
3.7.1.1 Member code check 57
3.7.1.2 Joint check 58
3.7.2 From the graphic window. *59*
3.7.2.1 Member code check 59
3.7.2.2 Joint check 61
3.7.3 From object property *62*
3.7.3.1 Detailed results for member check. 62
3.7.3.2 Detailed results for joint check. 64
3.8 MODIFY AND RE-RUN CODE CHECK 67
3.8.1 Change code checking parameters only *67*
3.8.2 Modify structural data without re-running analysis *68*
3.8.3 Modify structural data and re-run analysis *70*
3.9 MAKE A REPORT 71

4. BEAM STRESSES 73

4.1 RESULT SELECTION 75
4.1.1 Limitation on stress computation *77*
4.1.2 Select beams *78*
4.1.3 Select loadcases *78*
4.1.4 Select component *79*
4.1.5 Select hotspot position *79*
4.1.6 Deflection shape *81*
4.2 GRAPHS 84
4.3 MAXIMUM AND DETAILED VALUES 87

5. REDESIGN 89

5.1 INTRODUCTION 89
5.2 REDESIGN EXAMPLE 90

6. POST PROCESSING OF SUPERELEMENT RESULTS 101

6.1 MODELLING AND POST-PROCESSING 102
6.1.1 GeniE first level superelements *103*
6.1.1.1 Define the superelement number 103
6.1.1.2 Make the finite element model 104
6.1.1.3 Export the finite element model 104
6.1.1.4 Make the global load combinations – combine results in Prepost 105
6.1.1.5 Make the global load combinations – all combinations done in Presel 106
6.1.2 Post-processing *107*
6.2 PROCESS MODELLING IN BRIX EXPLORER 110
6.2.1 Make the flowchart *110*
6.2.2 Execute the flowchart *111*
6.2.2.1 Run GeniE 111
6.2.2.2 Run Presel 112
6.2.2.3 Run Sestra 112
6.2.2.4 Run Prepost 113
6.2.2.5 Re-use a GeniE activity to do post processing 113
6.2.2.6 Run Xtract 113

7. EXECUTION OF GENIE 115

7.1 PROGRAM ENVIRONMENT 115
7.2 FILES CREATED DURING CODE CHECKING 115

8. DIALOG DESCRIPTION 117

8.1 CODE CHECKING MENUS FROM THE BROWSER 117
8.1.1.1 Make a new capacity manager 117
8.1.1.2 Make capacity members 117

8.1.1.3 Create joints 117
8.1.1.4 Add a run 118
8.1.1.5 Generate code checking forces 119
8.1.1.6 Execute the code check 119
8.1.1.7 Set a code check run to active 119
8.1.1.8 Edit a code check description 119
8.1.1.9 Save a code check report 120
8.1.1.10 Delete 120
8.2 CODE CHECKING MENUS FROM THE GRAPHICAL WINDOW 120
8.2.1.1 Properties 121
8.2.1.2 Labels 121
8.2.1.3 Colour coding the results 121
8.3 CODE CHECKING MENUS FROM THE PULLDOWN MENU 122
8.3.1.1 Specify the colour coding of code check results 122
8.3.1.2 Specify the line width of capacity members 123

9. APPENDIX A: REFERENCES 125

10. APPENDIX B: CODE CHECKS AND NOMENCLATURE 127

10.1 THE CHECKS PERFORMED 127
10.2 NOMENCLATURE 128

11. APPENDIX C: CODE CHECKING BASED ON FRAMEWORK 129

11.1 IMPORT THE MODEL USING FINITE ELEMENT NUMBERING 132
11.2 GENERATE MEMBERS IN FRAMEWORK 132
11.3 IMPORT THE MODEL USING CONCEPT NAMES, NO SPLIT OF STRUCTURE 133
11.4 IMPORT THE CONCEPT MODEL, SPLIT STRUCTURE AT REINFORCED STRUCTURAL JOINTS 134
11.5 IMPORT THE CONCEPT MODEL, SPLIT STRUCTURE AT ALL STRUCTURAL JOINTS 135

This page is blank for printing

1. INTRODUCTION – CODE CHECKING OF BEAMS

This is the user manual for the part of GeniE dedicated to code checking of beams on offshore as well as onshore structures. In addition, stress calculations and presentations are covered.

This user manual assumes that the user has knowledge in the use of GeniE as covered by the GeniE User Manual Volume I and II – the main user manual and the one describing how to run analysis including wave loads and structure-pile-soil interaction.

This manual describes how to create a capacity model (or code checking model), how to perform the code checking as well as how to report the results.

Chapter 2 gives an overview while Chapter 3 contains a more detailed description on the code checking. Chapter 4 and 5 give examples on how to present beam stresses and how to do post processing of imported results from a super element analysis.

1.1 How to read this manual

Read Chapter 2 "*Beam code checking features of GeniE"* to get an overview on code checking features of GeniE (what you can do).

Read Chapter 3 "*How to do beam code checking"* to learn how to set up a capacity model, perform the code checks and how to report the results.

Read Chapter 4 and 5 "*Beam stresses"* and "Post-processing of super-element results" to learn how to document beam stresses as well as to import results from a super-element analysis to do post-processing like e.g. code-checking.

Read Chapter 6 *Execution of GeniE* for information on files associated with GeniE – focus is on the files generated during code checking.

Read Chapter 7 *Command Description* for a detailed documentation of relevant commands for code checking (pull down menus, tool buttons, and context sensitive menus).

A command from the menu list (also referred to as the pulldown menu) is written like this: ***Insert*|*Beam*|*Dialog***. The name of a tool button is written like this: **Basic plate**. A function buttons is referred to like this: **F1**.

GeniE comes with a context sensitive menu. You invoke this menu by pushing your right mouse button when the mouse is located above a selected object. In this manual this operation is termed **RMB**. The commands on the context sensitive menu are written like this: *Join Beams*.

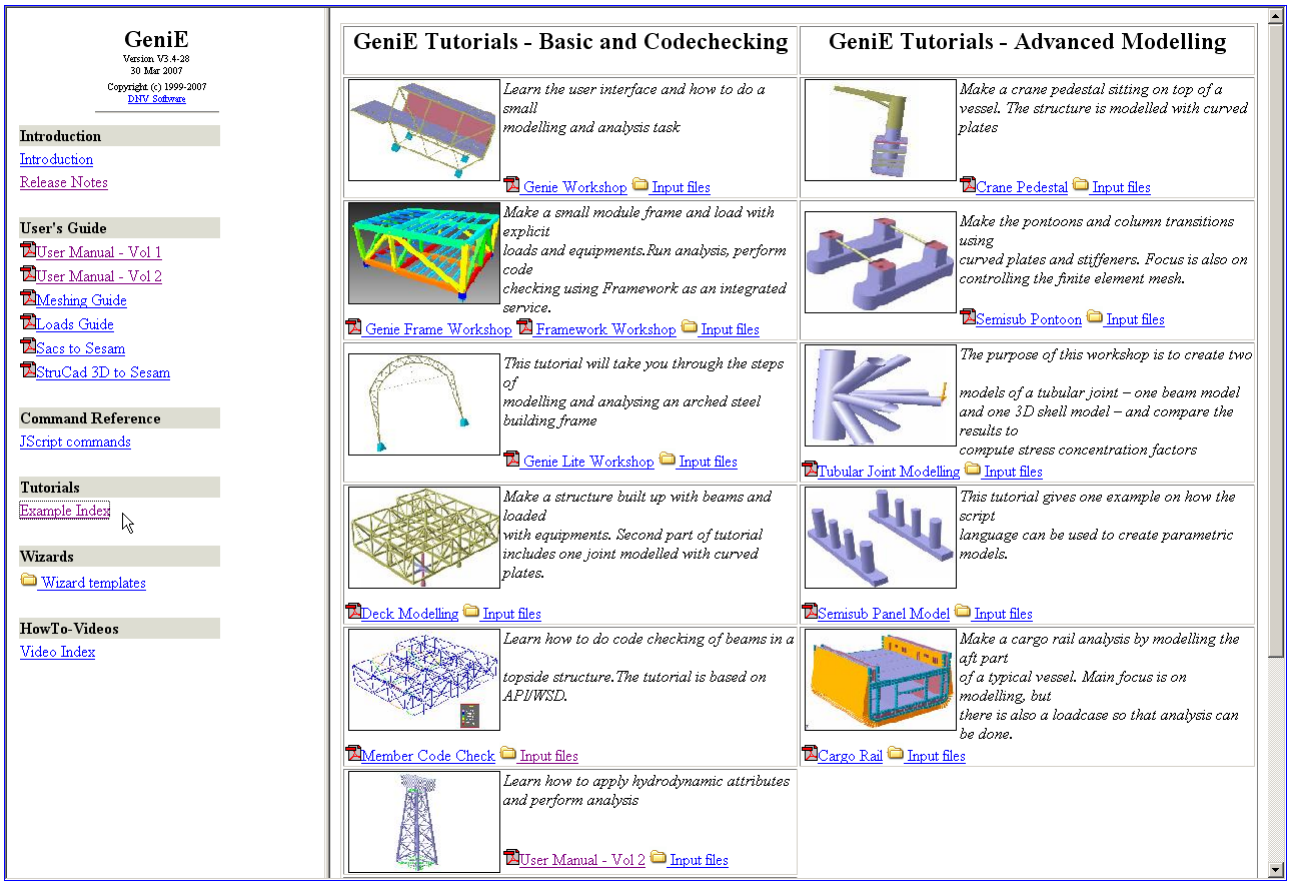
It is particularly noted that this User Manual[1] documents all capabilities of GeniE. If you do not have access to the program extension "*Code checking of beams (CCBM)*" there are several items in this manual you do not have access to in your program. These features are blanked out in your program version.

1.2 Learning from tutorials for code checking

GeniE comes with an on-line help system (***Help*|*Help Topics*** or **F1**). Its purpose is to provide easy access to release notes, limitations, tutorials, wizards and this user manual. In addition it contains a detailed documentation of all available commands in the journalling system (based on J-script). There are also videos showing how to do certain operations, these are best viewed using resolution 1280x1024.

The easiest way to find the tutorials is from GeniE's help page. For code checking the most relevant tutorials are called "Member code check" and "Jacket with tubular joints".

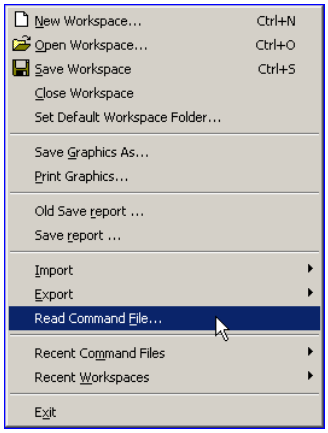
[1] Adobe Acrobat Reader version 8.0 is recommended.

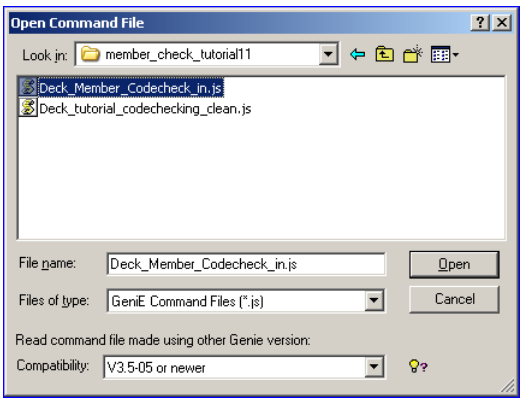

GeniE
Version V3.4-28
30 Mar 2007
Copyright (c) 1999-2007
DNV Software
Introduction
Introduction
Release Notes
User's Guide
User Manual - Vol 1
User Manual - Vol 2
Meshing Guide
Loads Guide
Sacs to Sesam
StruCad 3D to Sesam
Command Reference
JScript commands
Tutorials
Example Index
Wizards
Wizard templates
HowTo-Videos
Video Index
GeniE Tutorials - Basic and Codechecking
Learn the user interface and how to do a small modelling and analysis task
Genie Workshop Input files
Make a small module frame and load with explicit loads and equipments.Run analysis, perform code checking using Framework as an integrated service.
Genie Frame Workshop Framework Workshop Input files
This tutorial will take you through the steps of modelling and analysing an arched steel building frame
Genie Lite Workshop Input files
Make a structure built up with beams and loaded with equipments. Second part of tutorial includes one joint modelled with curved plates.
Deck Modelling Input files
Learn how to do code checking of beams in a topside structure.The tutorial is based on API/WSD.
Member Code Check Input files
Learn how to apply hydrodynamic attributes and perform analysis
User Manual - Vol 2 Input files
GeniE Tutorials - Advanced Modelling
Make a crane pedestal sitting on top of a vessel. The structure is modelled with curved plates
Crane Pedestal Input files
Make the pontoons and column transitions using curved plates and stiffeners. Focus is also on controlling the finite element mesh.
Semisub Pontoon Input files
The purpose of this workshop is to create two models of a tubular joint – one beam model and one 3D shell model – and compare the results to compute stress concentration factors
Tubular Joint Modelling Input files
This tutorial gives one example on how the script language can be used to create parametric models.
Semisub Panel Model Input files
Make a cargo rail analysis by modelling the aft part of a typical vessel. Main focus is on modelling, but there is also a loadcase so that analysis can be done.
Cargo Rail Input files

The most efficient way to work with the tutorials is to make a print-out of the tutorials, start GeniE, create a new workspace (command ***File|New workspace***) and follow the steps in the tutorial. Each tutorial comes with a pre-defined journal file (command file) – you find these from the help page as shown above under "Input Files". If you want to use these files the steps are as follows:

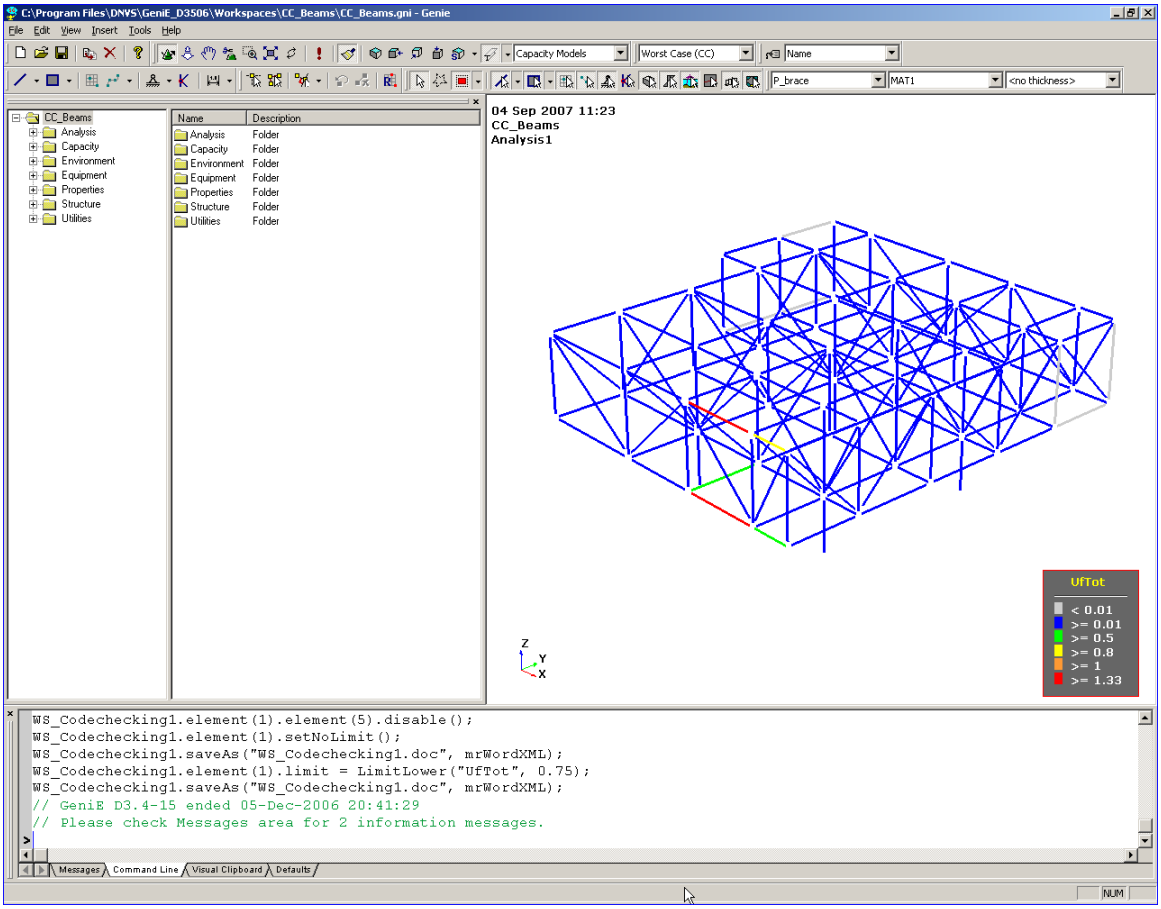
1. Create a new workspace ***File|New Workspace|<name>***
   (Keep the default settings for tolerant modelling and database units)
2. Read in the journal file from ***File|Read Command File***|<browse until you find the desired input file>
3. Save your model by ***File|Save***
4. You have now created the same model as in the tutorial you selected.

In the example below the <name> has been set to *CC_Beams* and the imported file is for the tutorial "Member code check".





The sequence above creates the following view in GeniE (the colour background has been set to white and the view is set to “Capacity Models”:



You may also read in a journal file by using drag-and-drop. Simply select a journal file from your browser and drop it into the command line window.

1.3 Acronyms frequently used in the User Manual

Acronym	Explanation
RMB	Right Mouse Button
LMB	Left Mouse Button
GUI	Graphical User Interface
DOF	Degree Of Freedom
CLI	Command Line Interface
LJF	Local Joint Flexibility
FEM file	Sesam Input Interface File – a formatted ASCII model/load file
SIN file	Sesam Interface NORSAM – a direct accessible result file
Brix Explorer	Brix Explorer configured for Sesam

1.4 Code checking on large models

The member code check result database is very big. For a large model it can reach several gigabytes in size. Earlier version including V5.3-10 have stored the results in memory and saved them to disk when you saved your workspace. This put a significant limitation to the number of capacity checks that could be stored in a single workspace. The memory limitation for the in-memory database is 2GB on a 64bit version of Windows.

From GeniE V5.4, the code check results are stored on disk in a file based database based on HDF5 technology. What this means is that you are able to code check much larger models than in the previous versions.

The code check results files are stored under

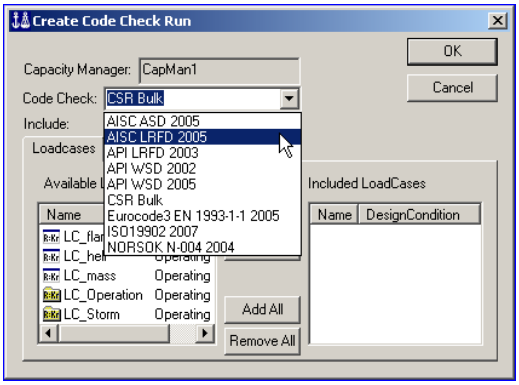
<Workspace Folder>/<CapacityManagerName>/<RunDescription>.h5

These files are part of the database, and must be included if you want to copy the workspace folder.

2. BEAM CODE CHECKING FEATURES OF GENIE

Code checking of beams based on the results from a linear structural analysis can be done in GeniE. The features include checks against allowable stress levels, member stability and punching shear of tubular joints.

You should be familiar with the relevant rules and procedure of the type of code checking you want to do as this user manual is not intended to cover such. For example, if you want to do a code checking according to the API rules you should know this code of practice.



2.1 Code checking standards

GeniE supports the following code checking standards:

- *API-WSD 2002 – Offshore structures*
  For tubular profiles: American Petroleum Institute RP 2A-WSD (21st edition December 2000, Errata and Supplement 1, December 2002)
  For non-tubular profiles: American National Standard; Specification for Structural Steel Buildings", March 9, 2005
- *API-WSD 2005 – Offshore structures*
  For tubular profiles: American Petroleum Institute RP 2A-WSD (21st edition December 2000, Errata and Supplement 2, October 2005)
  For non-tubular profiles: American National Standard; Specification for Structural Steel Buildings", (March 9, 2005)
- *API-LRFD 2003 – Offshore structures*
  For tubular profiles: American Petroleum Institute LRFD (1st Edition/July 1, 1993/ Reaffirmed, May 16, 2003)
  For non-tubular profiles: American National Standard; Specification for Structural Steel Buildings", (March 9, 2005)
- *NORSOK 2004 - Offshore structures*.
  For tubular profiles: NORSOK STANDARD N-004, Rev. 2, October 2004, Design of steel structures
  For non-tubular profiles: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian National Annex, 2006
- *ISO 19902 2007 - Offshore structures*.
  For tubular profiles: INTERNATIONAL STANDARD ISO 19902, Petroleum and natural gas industries — Fixed steel offshore structures (First edition 1 December 2007)
  For non-tubular profiles: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian National Annex, 2006
- *AISC 2005 – Onshore structures*
  For tubular and non-tubular profiles: American National Standard; Specification for Structural Steel Buildings", March 9, 2005. The check covers design/utilisation of members according to the provisions for *Load and Resistance Factor Design* (LRFD) or to the provisions for *Allowable Strength Design* (ASD).
- *EUROCODE 3 – Onshore structures*
  For tubular and non-tubular profiles: EUROCODE 3, EN 1993 Part 1-1: General rules and rules for buildings. It is also possible to select the preferences according to the Norwegian National Annex, 2006.
- *DANISH STANDARD 412 / 449 – Onshore and Offshore structures*
  For tubular profiles only in both DS 412 and DS 449

For references to each code check standard, please consult Appendix A. Code checking using older code check revisions may also be done using the Sesam Framework module. For further references, please consult Appendix C.

2.2 Available code checks

The code checks available are as follows:

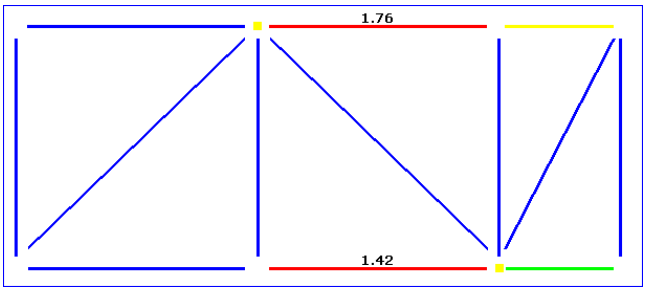
• Member check

• Hydrostatic collapse

• Punching shear

• Conical transition

A member check of a frame structural member is performed to assess whether the member is subjected to acceptable stress levels. This check is performed through the use of the equations presented in the various code checking standards. These equations deliver results – the usage factor – according to capacity of the cross-sections and capacity taking into account the potential failure due to buckling phenomena. If this usage factor is less than 1.0 then the member is regarded to be "safe". If the usage factor is greater than 1.0 then the member is "overloaded" and this is highlighted by the program. A member check is by default performed at five positions; at the two ends of the member, the midpoint and at the quart positions. In addition, additional code checking positions are determined based on variations in section profiles or materials (like in a segmented member) or where the maximum moments (in-plane and out-of- plane) occur. This means that the code checking positions may vary from load case to load case.



A hydrostatic collapse check is performed to assess the member induced stresses due to the action of hydrostatic pressure and other externally applied loads. This check is for relevant standards integrated with the member check (see next page).

A punching shear check is performed on the brace member at a joint to assess the shear through the chord. As for the other checks this assessment is made through the use of a 'punching shear interaction equation' which delivers a usage factor. The punching shear check is performed for all braces at selected joints.

A cone check is performed to assess the stresses in the transition between cone and cylinder. As for the other checks this assessment is made through the use of a 'conical transition interaction equation' which delivers a usage factor. The cone check is performed for cylinder and cone at both ends of the conical transition. This check is for relevant standards integrated with the member check (see next page).

Section types that may be code checked are:

- Tubular sections (PIPE)
- Symmetrical/un-symmetrical I or H sections (I)
- Channel sections (CHAN)
- Box sections (BOX)
- Massive bar sections (BAR)
- Angle sections (L)
- General sections (GENE)

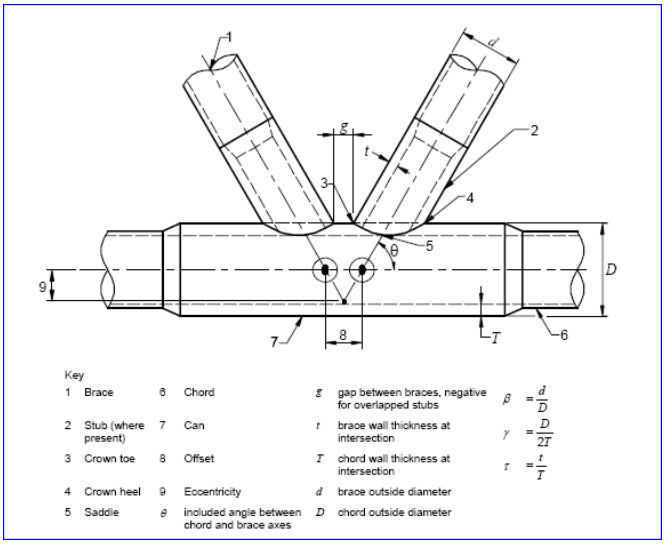
The table below shows the type of check that may be performed for each code of practice and the section type that may be processed.

Code of practice	**Check**	**Member section**						
		PIPE	I	CHA	BOX	BAR	L	GEN
API-WSD 2002 including AISC ASD 2005	Member	API	AISC	AISC	AISC	AISC	AISC	AISC
	Punching shear	API						
	Hydrostatic collapse	API						
	Conical transition	API						
API-WSD 2005 including AISC ASD 2005	Member	API	AISC	AISC	AISC	AISC	AISC	AISC
	Punching shear	API						
	Hydrostatic collapse	API						
	Conical transition	API						
API-LRFD 2003 including AISC LRFD 2005	Member	API	AISC	AISC	AISC	AISC	AISC	AISC
	Punching shear	API						
	Hydrostatic collapse	API						
	Conical transition	API						
NORSOK 2004 including EUROCODE 3 (EN 1993-1-1)	Member	NOR	EUR	EUR	EUR	EUR	EUR	EUR
	Punching shear	NOR						
	Hydrostatic collapse	NOR						
	Conical transition	NOR						
ISO 19902 1997 including EUROCODE 3 (EN 1993-1-1)	Member	ISO	EUR	EUR	EUR	EUR	EUR	EUR
	Punching shear	ISO						
	Hydrostatic collapse	ISO						
	Conical transition	ISO						
AISC ASD 2005	Member	AISC	AISC	AISC	AISC	AISC	AISC	AISC
AISC LRFD 2005	Member	AISC	AISC	AISC	AISC	AISC	AISC	AISC
EUROCODE 3 (EN 1993-1-1)	Member	EUR	EUR	EUR	EUR	EUR	EUR	EUR
DANISH STANDARD 412/449	Member	DS412 DS449						

2.3 Model requirements

When modelling the structure some of the data needed for the punching shear code check is defined. These are (for references please consult User Manual Vol. I under *Tubular joint modelling*):

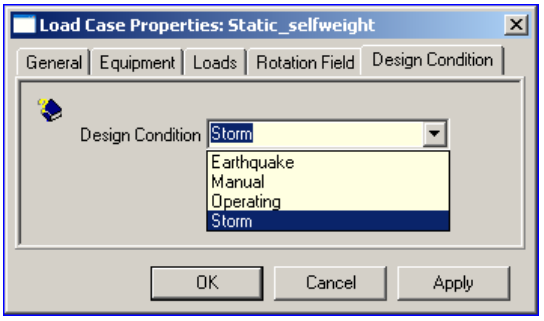
- Modelling of tubular joints including:
  - Automatic (or explicit) definition of CAN reinforcement assigned to a chord
  - Automatic (or explicit) definition of STUB reinforcement assigned to a brace
  - Automatic (or explicit) definition of CONES assigned to transitions between CAN/CHORD and STUB/BRACE
  - Automatic (or explicit) assignment of plane-wise gaps



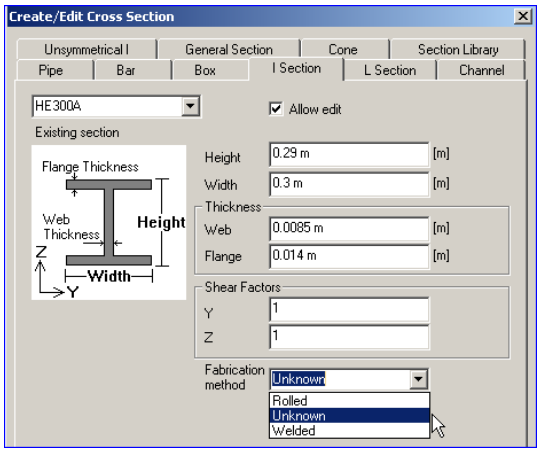
- Definition of different yield strength at different parts of the structure to account for differences in the grade of steel. Yield strength is defined through a material property

For a member code check it is necessary to assign the buckling length – or by use of effective length factors – to a member to consider in-plane and out-of-plane buckling effects. In addition it is possible to account for secondary moments due to axial loads in buckling calculations by explicitly assignment of moment amplification factors or by using factors as recommended by the various codes of practices. The other code specific code check parameters are described in Appendix B.

Load combinations may be defined before or after finite element analysis. When using the API-WSD code checks you should denote the loadcases with design conditions “Operating”, “Storm” or “Earthquake” to automatically increase the allowable stresses (0%, 33.3% and 70% respectively). API-LRFD and ISO 19902 use this design condition to specify the load factor for hydrostatic pressure.



The AISC code includes safety factors for fabrication method. Notice that all cross sections are as default set to fabrication status “Unknown”. Unknown is in the code check treated as Built-up (Welded). Cross sections imported from the section libraries are also set to status “Unknown”.



2.4 Code checking procedure

A typical procedure adopted for a member and/or punching shear code check analysis may be as follows:

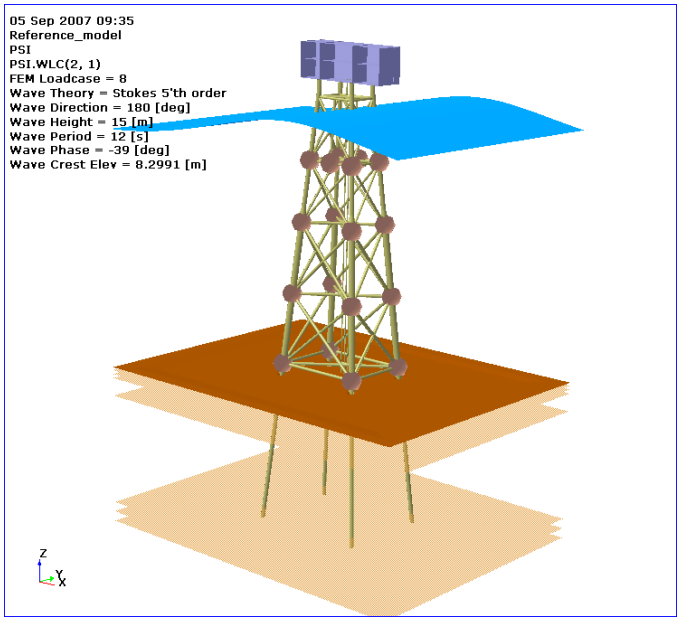
- Modelling of material, sections, structure, loads and boundary conditions. If relevant, the modelling also comprises the environment (wind, waves and current), the piles and the soil.
- If you want to perform a punching shear of tubular joints you need to insert joints and complete the tubular joint modelling (can, stub, cone, plane-wise gap).
- Run the finite element analysis. If the structure is subjected to water and has pile/soil it is necessary to include wave load analysis and pile/soil/structure analysis. These options are pre-defined activities you can select from when setting up your analysis.
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager(s). You decide which analysis you want to base the code checking on. You may have several capacity managers – each capacity manager may have one set of member and joints. These may be selected from the whole structure or from named sets
- Define the members and joints. When defining the members the global default buckling length of each member is assigned. For members you define how to split continuous concept members into code checking members.
- Create a code check run. The purpose of this task is to decide which code of practice to use, which loadcases to include and to specify other global factors like e.g. moment amplification factors or other safety factors. You also specify what you want to do - member code check and/or punching shear check
- Assign local details to individual beams, typically another buckling length compared to the rest of the model.
- Compute the code checking forces. These are computed at pre-defined positions (at ends, in middle and at quarter positions), at positions where material or section properties change (like in segmented beams) or where maximum in-plane and out-of-plane moments occur.
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify sections, materials or other code checking parameters and re-run.
- Make a report using the report generator and include pictures to it. The granularity of the report depends on how much details you want to add to it. There are several filters you can use to decide the content.

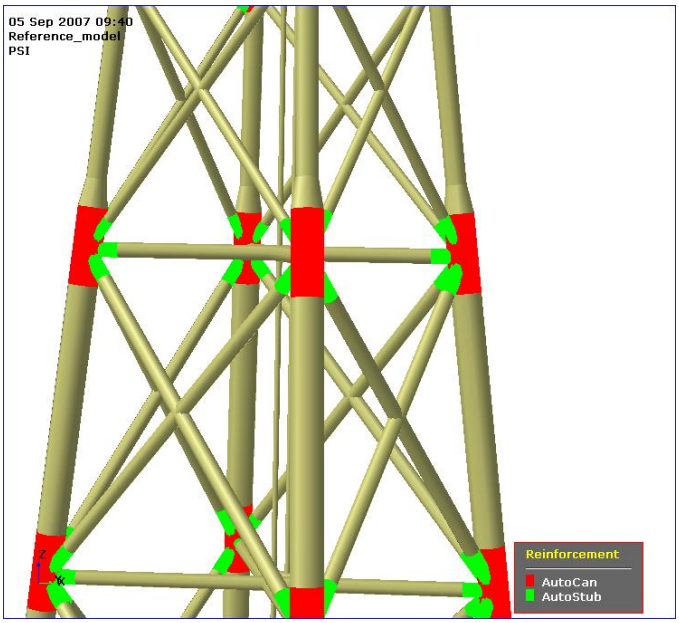
The above procedural steps from creating the capacity manager are illustrated in the following using the tutorial “Jacket with tubular joints”. A detailed description of each step is explained in the next Chapter.

2.4.1 A reference model

A reference model has been created by reading in the journal file part of the tutorial "Jacket with tubular joints". The purpose of this tutorial is to create a capacity model to do member and punching shear code checks, hence the model has tubular joints with cans, stubs, cones and gaps. The reference model has one analysis activity called "PSI".

The red balls indicate where joints have been defined – these are also the positions where the punching shear will be performed. The colour coding indicates the cans and the stubs.


05 Sep 2007 09:35
Reference_model
PSI
PSI.WLC(2, 1)
FEM Loadcase = 8
Wave Theory = Stokes 5'th order
Wave Direction = 180 [deg]
Wave Height = 15 [m]
Wave Period = 12 [s]
Wave Phase = -39 [deg]
Wave Crest Elev = 8.2991 [m]

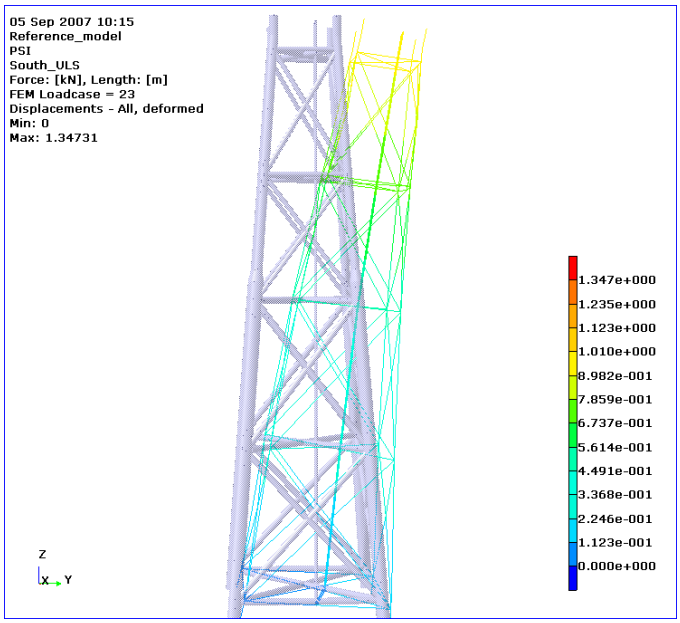


2.4.2 Making load combinations

The reference model has a number of basic loadcases (manually applied as well as wave loads) and load combinations. In this case, four load combinations have been made after the analysis (North_ULS, East_ULS, South_ULS, West_ULS) by combining other loadcases or load combinations and apply load factors to each of them. Each of the four load combinations have been assigned design condition "Storm".

The picture to the right shows displacements for the jacket part for load combination South_ULS.

As can be seen, the deformed shape may be displayed on top of the un-deformed shape (or alone), for quick evaluation of displacement results.

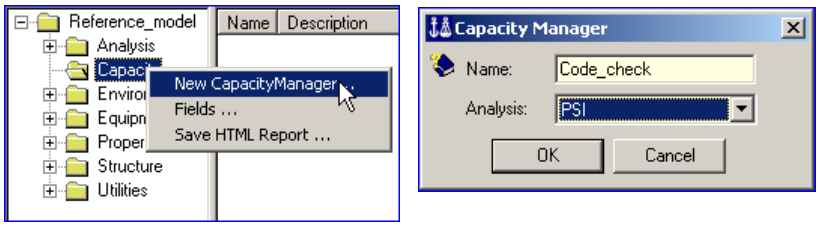


2.4.3 Create a capacity manager

The purpose of a capacity manager is to decide which analysis results to use in the code checking. You may have several capacity managers if you want to use

- various analysis results
- different selection of capacity members or joints

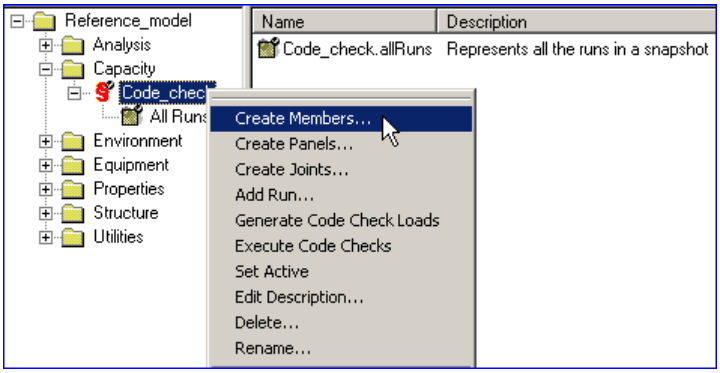
A capacity manager is created from the browser, and in this case it is given the name “Code_check”.

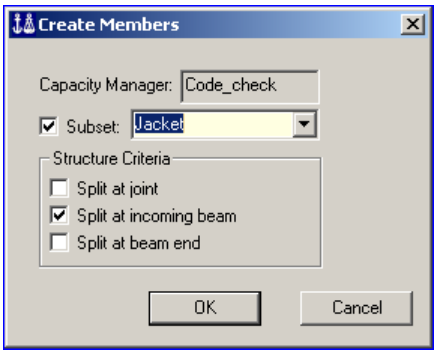


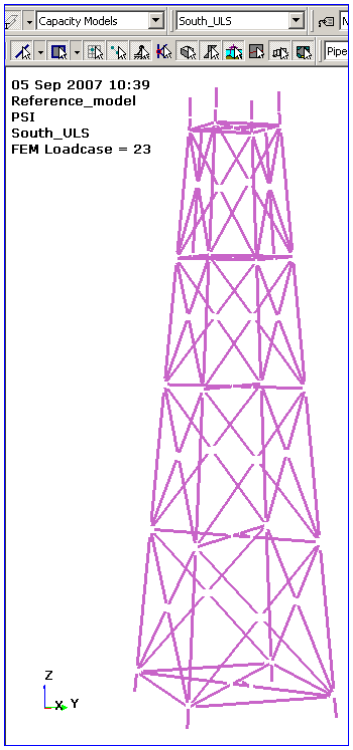
2.4.4 Create capacity members and joints

Concept modelling allows for definition of continuous members across incoming beams and joints. In a code check the members are normally not the same as the concept model, hence it is necessary to define the capacity members by splitting the concept members. In this case the capacity members are defined by splitting the concept model where the option “Split at incoming beams” has been used. Furthermore, the members are created for the jacket part only.

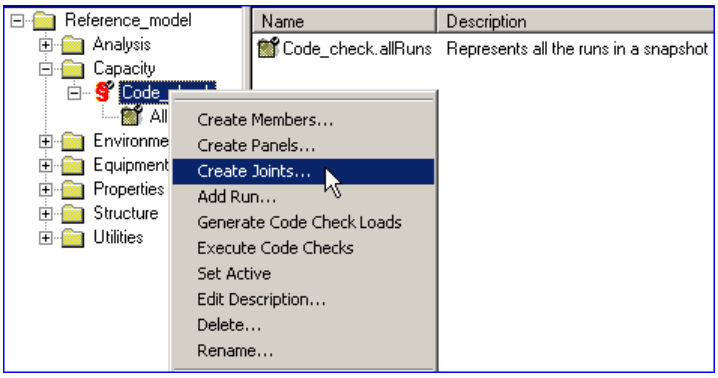
The capacity members can be visualised by using the view setting Capacity Model

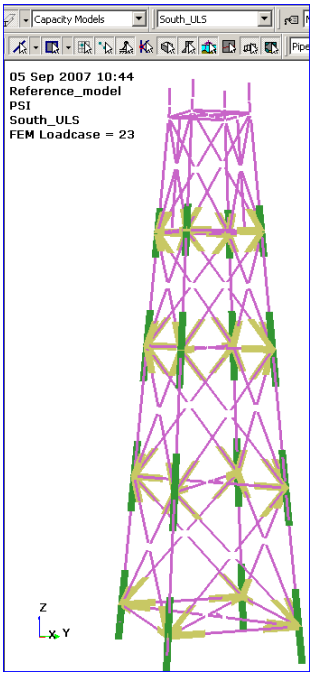


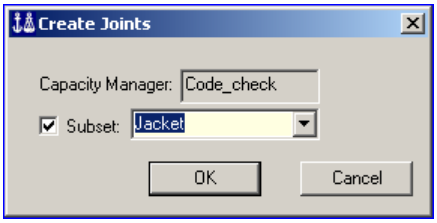




The capacity joints are defined similarly and the capacity model now contains the joints:



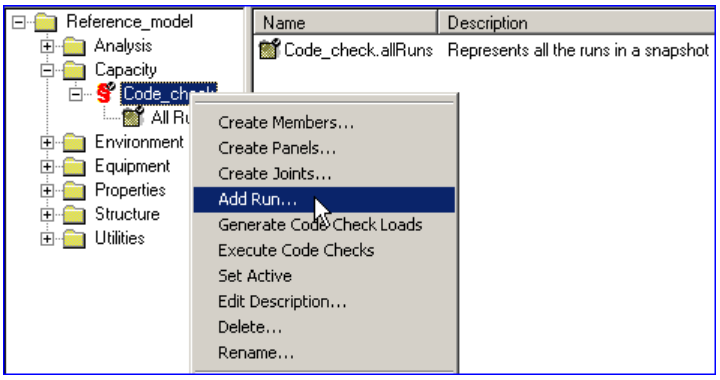




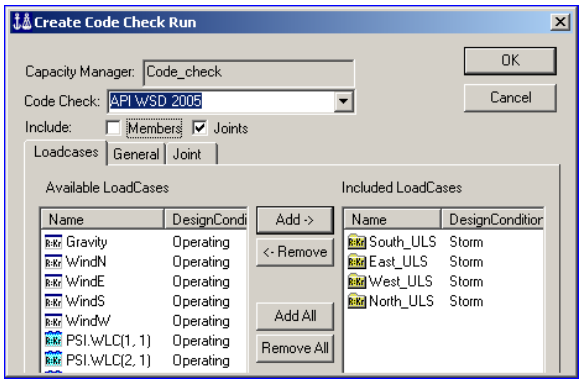
2.4.5 Create code check runs

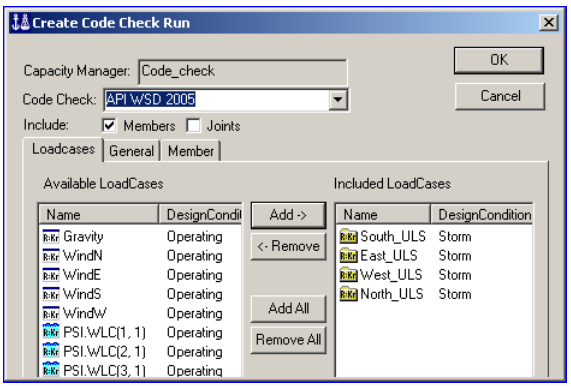
When you define a code check run you decide which code of practice, member and/or punching shear code check and global code checking parameters, i.e. valid for the complete capacity model (these may be overruled by local parameters).

In this case two different code check runs are defined; one for member code check only and the other for the punching shear.

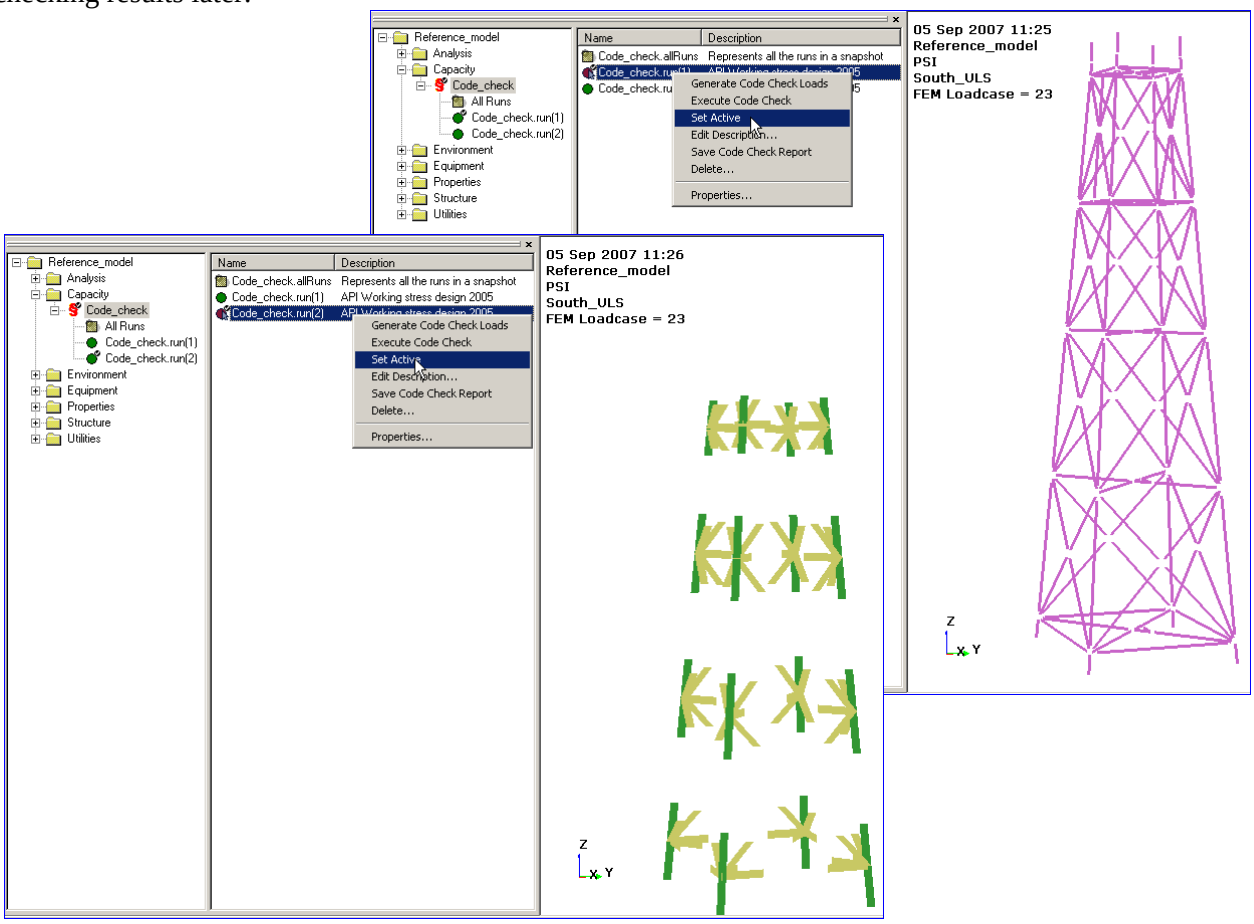


The code check runs for members and joints assume using API WSD 2005 and four loadcases as indicated. All other parameters are using program defaults. You learn how to modify the defaults in the next Chapter.





The code check runs are now available in the browser – the browser will be used when looking at the code checking results later.



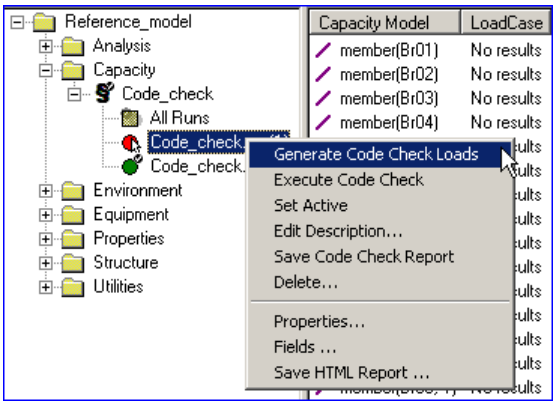
2.4.6 Compute code checking forces

The finite element analysis will compute forces and moments in the ends of each finite element (and the middle node if 2nd order straight beam elements are used). For the code checking it is necessary to compute the forces for other positions. The code checking positions are automatically determined by GeniE as follows:

- At beam ends, at quarter and middle positions
- At positions where there is a change in section or material properties like in a segmented member
- At positions where maximum in-plane and out-of-plane moments occur

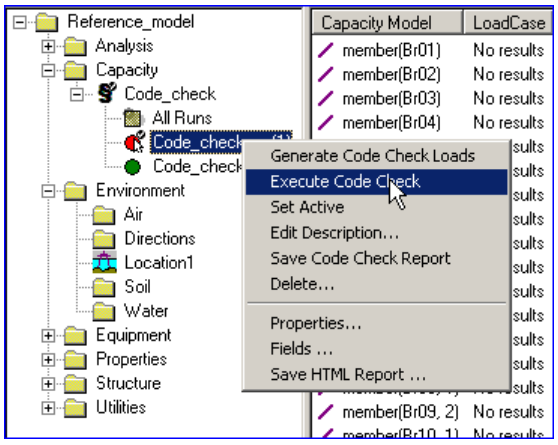
This means that the positions may vary from load case to load case.

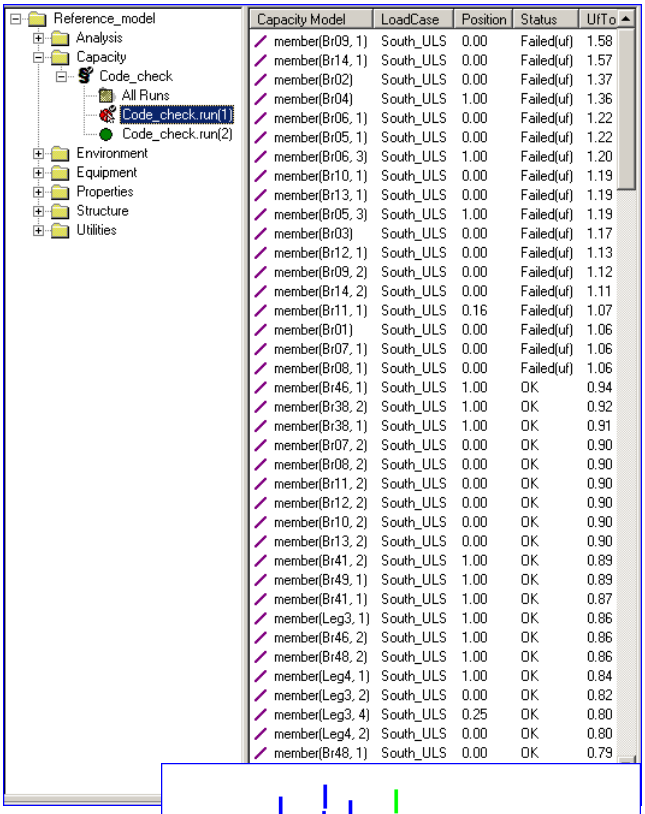
The code checking forces are computed from the browser and it must be done for all code check runs.



2.4.7 Perform the code check

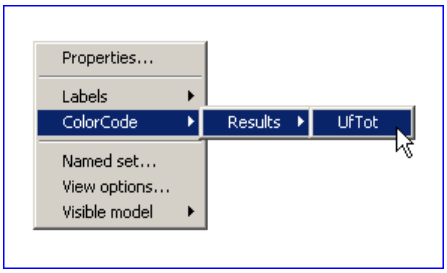
The code checks for member and punching shear can now be done from the browser (remember to specify which code check run is active). Below is shown how to start the code check and the associated results in the browser and a colour coded view.

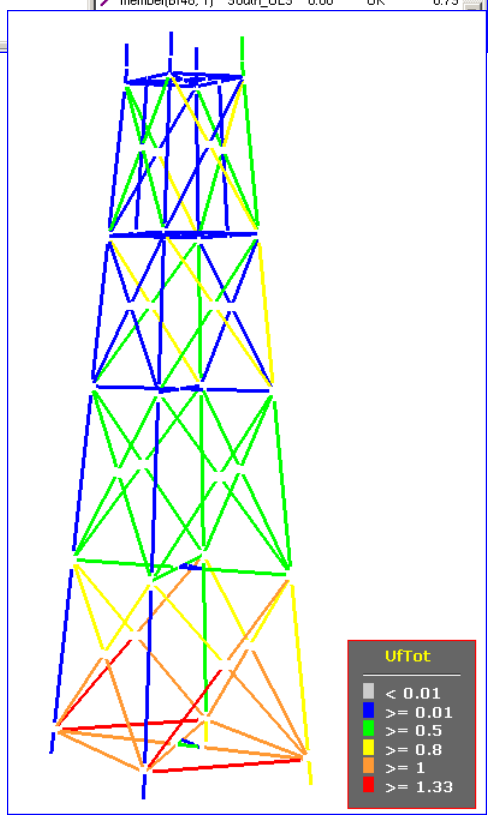



Reference_model
Analysis
Capacity
Code_check
All Runs
Code_check.run(1)
Code_check.run(2)
Environment
Equipment
Properties
Structure
Utilities
Capacity Model | LoadCase | Position | Status | UfTo
member(Br09, 1) South_ULS 0.00 Failed(uf) 1.58
member(Br14, 1) South_ULS 0.00 Failed(uf) 1.57
member(Br02) South_ULS 0.00 Failed(uf) 1.37
member(Br04) South_ULS 1.00 Failed(uf) 1.36
member(Br06, 1) South_ULS 0.00 Failed(uf) 1.22
member(Br05, 1) South_ULS 0.00 Failed(uf) 1.22
member(Br06, 3) South_ULS 1.00 Failed(uf) 1.20
member(Br10, 1) South_ULS 0.00 Failed(uf) 1.19
member(Br13, 1) South_ULS 0.00 Failed(uf) 1.19
member(Br05, 3) South_ULS 1.00 Failed(uf) 1.19
member(Br03) South_ULS 0.00 Failed(uf) 1.17
member(Br12, 1) South_ULS 0.00 Failed(uf) 1.13
member(Br09, 2) South_ULS 0.00 Failed(uf) 1.12
member(Br14, 2) South_ULS 0.00 Failed(uf) 1.11
member(Br11, 1) South_ULS 0.16 Failed(uf) 1.07
member(Br01) South_ULS 0.00 Failed(uf) 1.06
member(Br07, 1) South_ULS 0.00 Failed(uf) 1.06
member(Br08, 1) South_ULS 0.00 Failed(uf) 1.06
member(Br46, 1) South_ULS 1.00 OK 0.94
member(Br38, 2) South_ULS 1.00 OK 0.92
member(Br38, 1) South_ULS 1.00 OK 0.91
member(Br07, 2) South_ULS 0.00 OK 0.90
member(Br08, 2) South_ULS 0.00 OK 0.90
member(Br11, 2) South_ULS 0.00 OK 0.90
member(Br12, 2) South_ULS 0.00 OK 0.90
member(Br10, 2) South_ULS 0.00 OK 0.90
member(Br13, 2) South_ULS 0.00 OK 0.90
member(Br41, 2) South_ULS 1.00 OK 0.89
member(Br49, 1) South_ULS 1.00 OK 0.89
member(Br41, 1) South_ULS 1.00 OK 0.87
member(Leg3, 1) South_ULS 1.00 OK 0.86
member(Br46, 2) South_ULS 1.00 OK 0.86
member(Br48, 2) South_ULS 1.00 OK 0.86
member(Leg4, 1) South_ULS 1.00 OK 0.84
member(Leg3, 2) South_ULS 0.00 OK 0.82
member(Leg3, 4) South_ULS 0.25 OK 0.80
member(Leg4, 2) South_ULS 0.00 OK 0.80
member(Br48, 1) South_ULS 0.00 OK 0.79

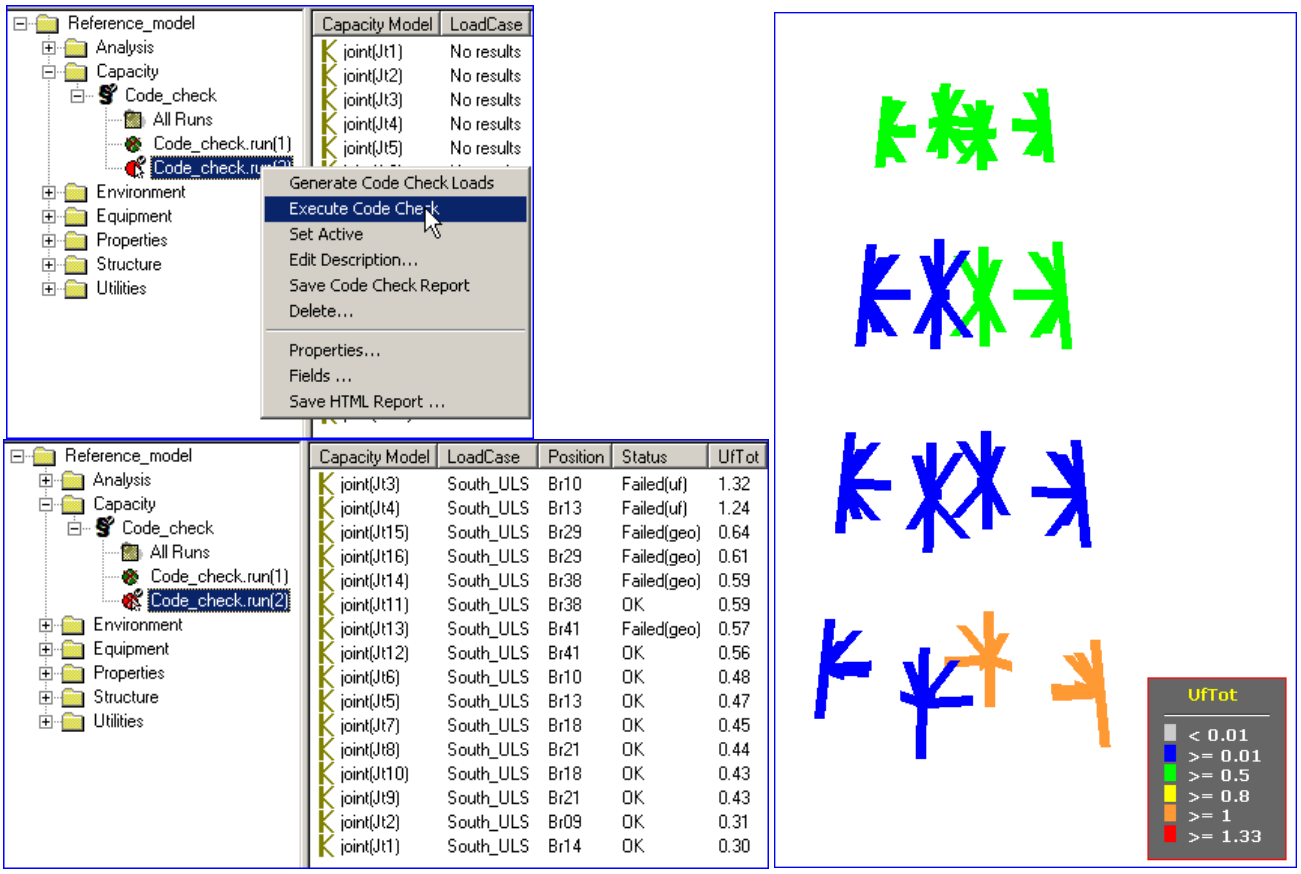
If you have several code check runs, you may compute the forces and execute the code checks for all your runs from the "All Runs" folder.

To make a colour coded view of the utilisation factors, simply select all capacity members, RMB and select *ColorCode|Results|UfTot*.





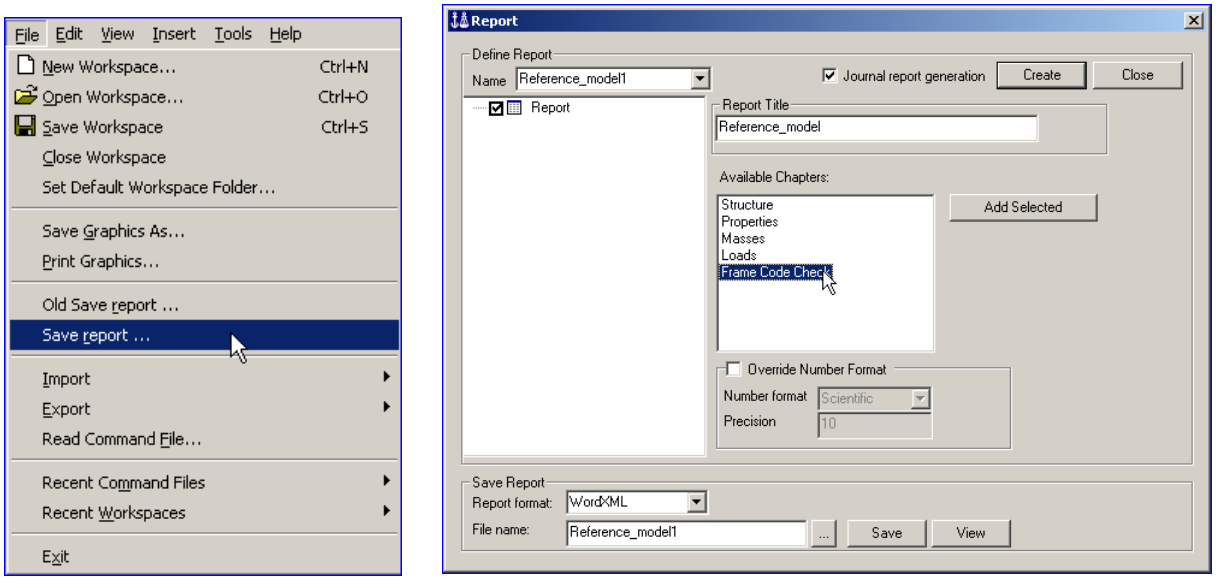
Similarly for the punching shear check:



The next Chapter describes in detail how you can interrogate each result.

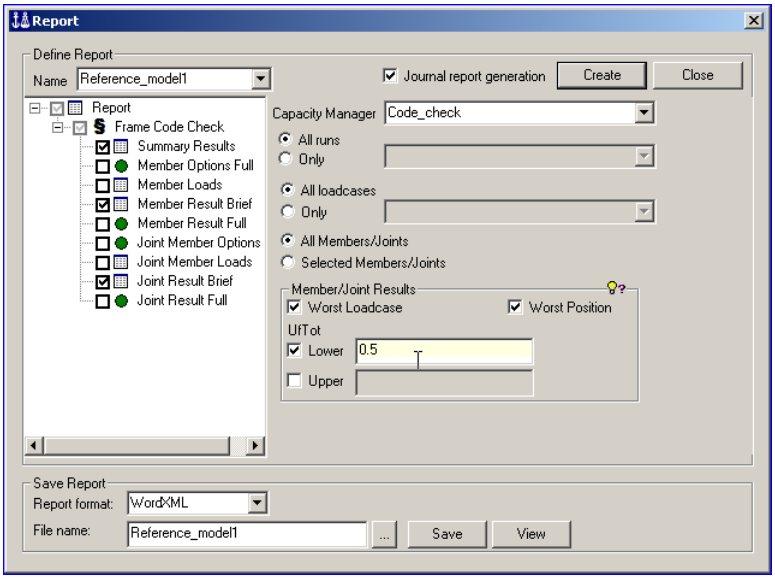
2.4.8 Making a code checking report

The report generator comes with templates to easily generate reports for viewing in text format (typically by using MS Notepad), html format (typically from an internet browser), spreadsheet (MS Excel) or formatted report (MS Word).You may use filters to limit the amount of data that is reported. Furthermore, your report settings may be saved so that you easily can recreate the same report in another code checking run.

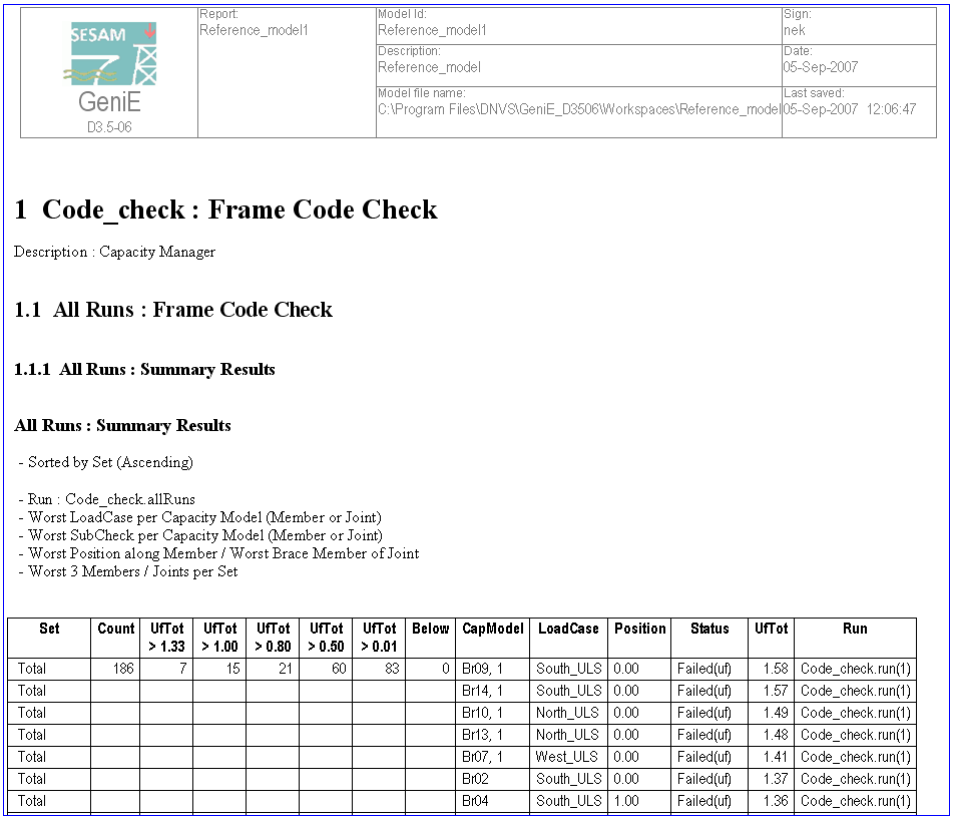


When you have added the Chapter “Frame Code Check” to your report it is possible to use the filters to specify the content you want.

This example uses the default report settings except for the limit of 0.5 for the lower utilisation factor.



A typical page of a report when viewing it in MS Word is shown to the right.



SESAM GeniE D3.5-06

Report:	Model Id:	Sign:
Reference_model1	Reference_model1	nek
	Description: Reference_model	Date: 05-Sep-2007
	Model file name: C:\Program Files\DNVS\GeniE_D3506\Workspaces\Reference_model	Last saved: 05-Sep-2007 12:06:47

1 Code_check : Frame Code Check

Description : Capacity Manager

1.1 All Runs : Frame Code Check

1.1.1 All Runs : Summary Results

All Runs : Summary Results

- Sorted by Set (Ascending)

- Run : Code_check.allRuns
- Worst LoadCase per Capacity Model (Member or Joint)
- Worst SubCheck per Capacity Model (Member or Joint)
- Worst Position along Member / Worst Brace Member of Joint
- Worst 3 Members / Joints per Set

Set	Count	UfTot > 1.33	UfTot > 1.00	UfTot > 0.80	UfTot > 0.50	UfTot > 0.01	Below	CapModel	LoadCase	Position	Status	UfTot	Run
Total	186	7	15	21	60	83	0	Br09, 1	South_ULS	0.00	Failed(uf)	1.58	Code_check.run(1)
Total								Br14, 1	South_ULS	0.00	Failed(uf)	1.57	Code_check.run(1)
Total								Br10, 1	North_ULS	0.00	Failed(uf)	1.49	Code_check.run(1)
Total								Br13, 1	North_ULS	0.00	Failed(uf)	1.48	Code_check.run(1)
Total								Br07, 1	West_ULS	0.00	Failed(uf)	1.41	Code_check.run(1)
Total								Br02	South_ULS	0.00	Failed(uf)	1.37	Code_check.run(1)
Total								Br04	South_ULS	1.00	Failed(uf)	1.36	Code_check.run(1)

3. HOW TO DO BEAM CODE CHECKING

This Chapter will guide you through the steps which are necessary to do code member code checking or a punching shear check. Two reference cases will be used; one topside model (for member code check) and a jacket (for punching shear check).

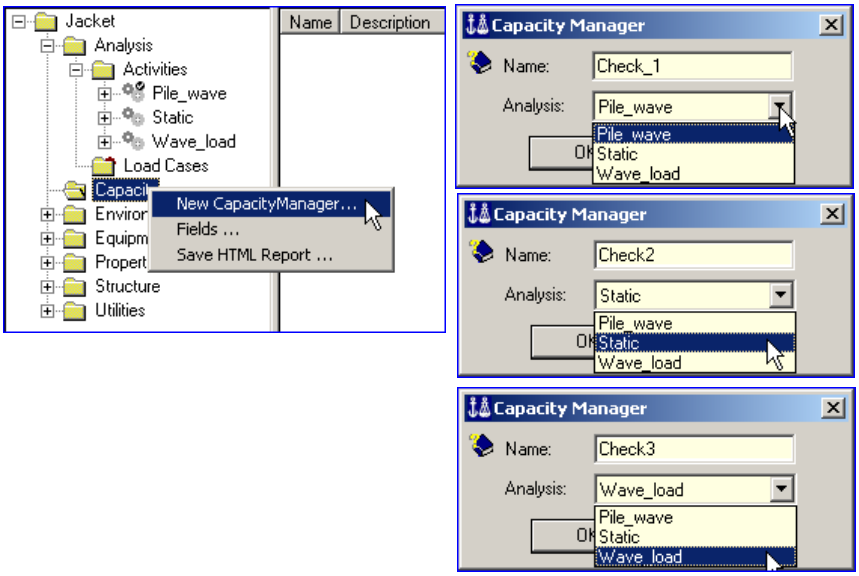
The following procedure may be used when performing code checks; each is described in detail except for the four first steps (please consult the User Manual Vol. I or III for guidance).

- Make a frame model. Plates may be included – they are disregarded in the code check.
- Perform tubular joint modelling if you want to do a punching shear check.
- Run the finite element analysis
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager
- Define the members and joints
- Create a code check run
- Assign specific settings to individual members and joints
- Compute the code checking forces
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify sections, materials or other code checking parameters and re-run.
- Make a report

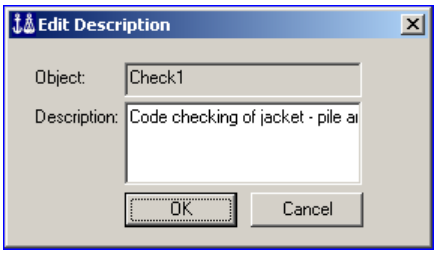
3.1 Create a capacity manager

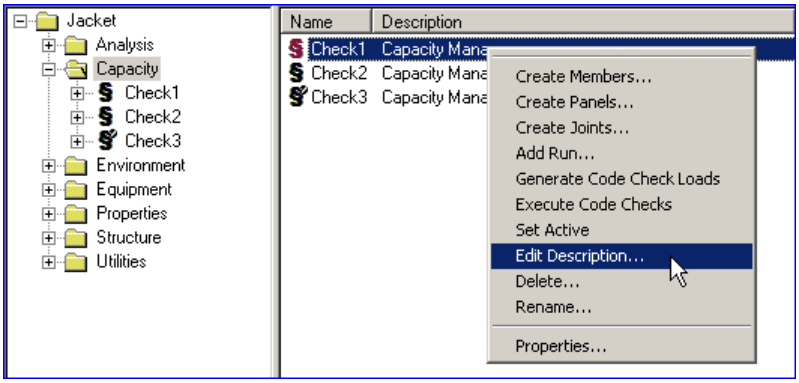
The purpose of a capacity manager is to decide which analysis results to use in the code checking. It is possible to have several analysis activities in GeniE where you can have different loadcases as well as analyse subsets of your model. To be able to do code checking you need to define multiple capacity managers referring to the different analysis in question.

A capacity manager is created from the browser, and in this case three capacity managers are defined to account for three different analyses.



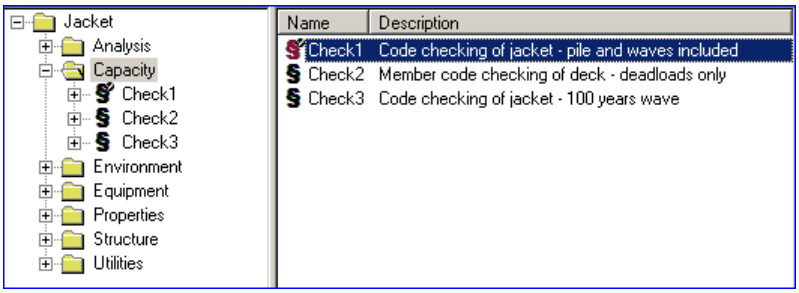
You may add additional description to each of the capacity managers.





The description is also shown in the browser.

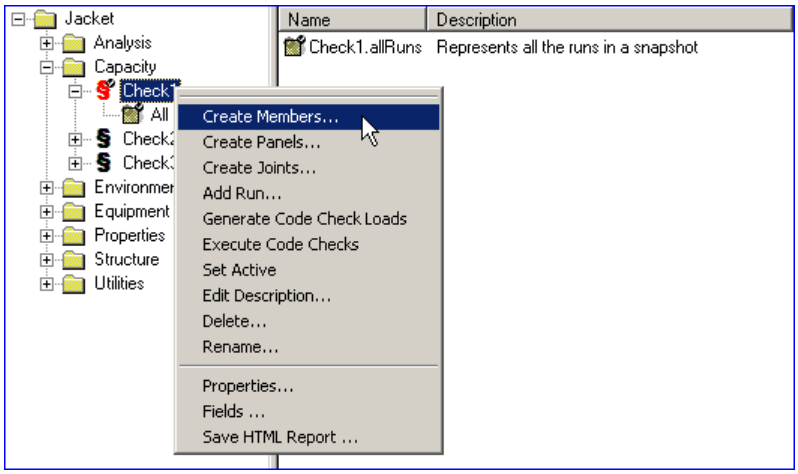
When making a report, the descriptions are also documented.

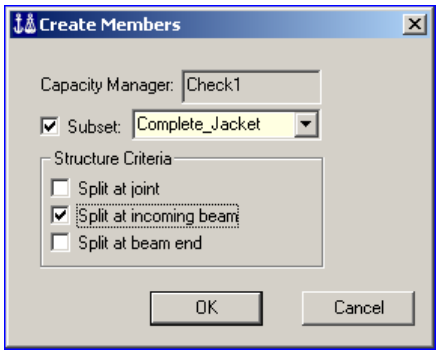


3.2 Define member and joints

When modelling a concept model it is possible to make continuous members that span several joints. This means that the concept model is different from a member capacity model which normally spans between two joints only. It is therefore necessary to split up the concept model into a member capacity model.

When the model is split or kept, the default buckling lengths are set since they are the same as the length of a capacity member.

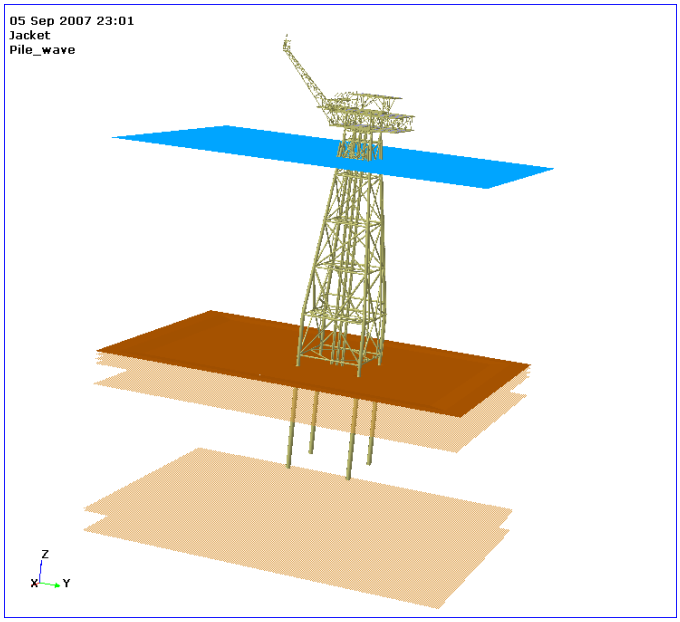


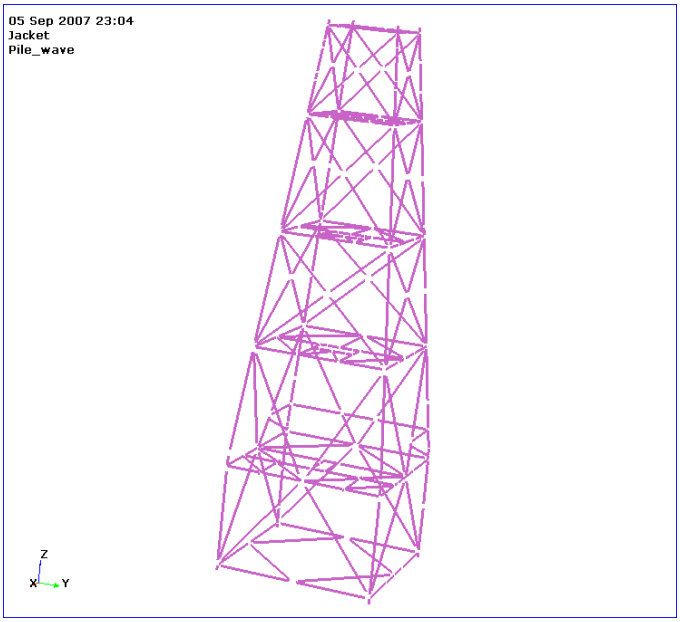


In this example the subset “Complete_Jacket” will be subject to the code checking. Furthermore, the continuous beams are split into capacity members where there are incoming beams to the continuous beam.

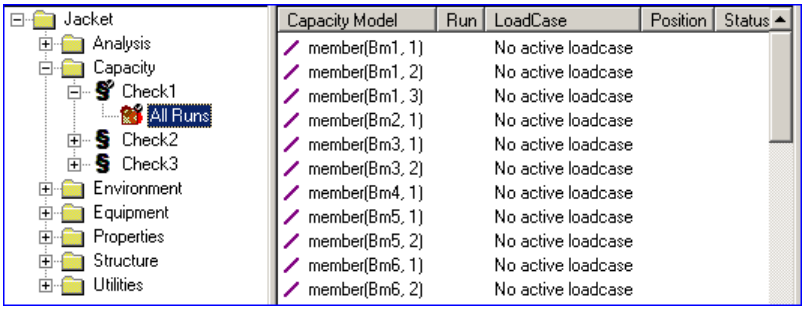
The naming convention of capacity members refer to the beams. If beam *BM25* is split into two capacity members they are denoted *member(BM25,1)* and *member(BM25,2)*.

The pictures show the concept model and the capacity members.

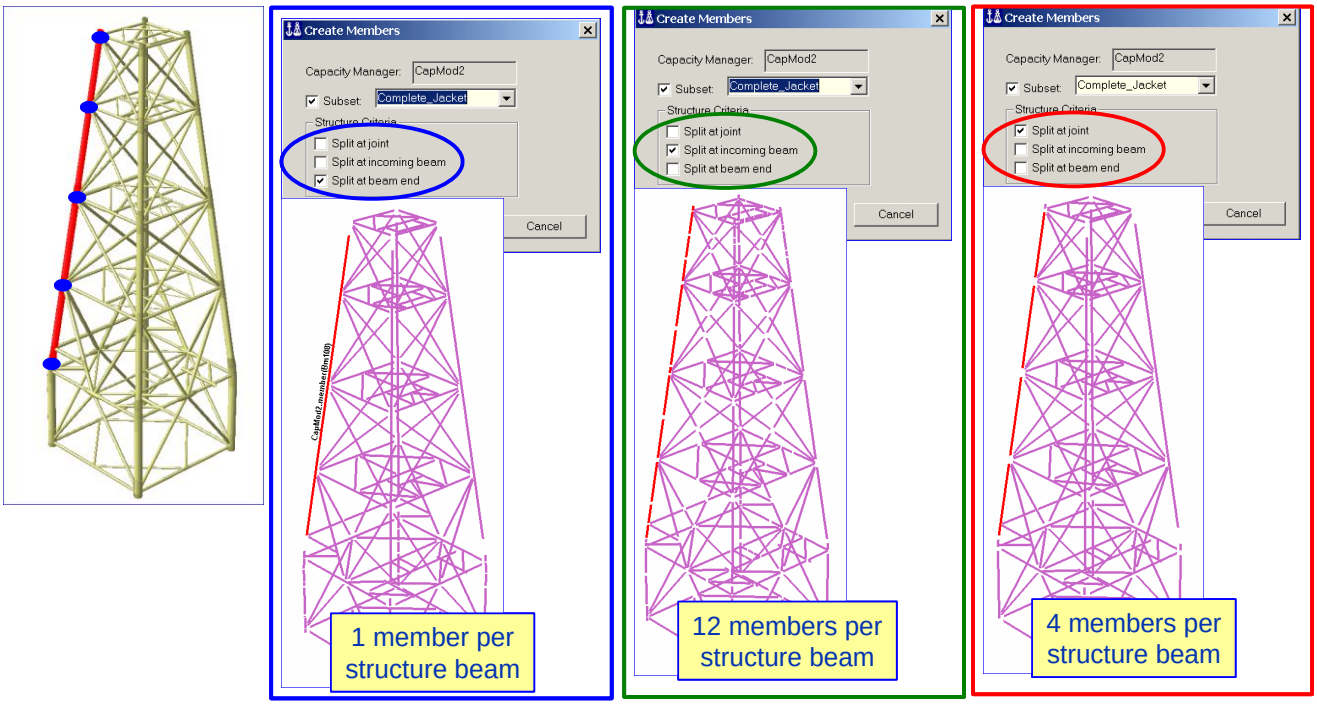




The browser lists each capacity member. If you want to navigate between the various capacity managers you need to specify which manager is active (select a manager, RMB and choose *Set Active*). In this case "Check1" is set to active.

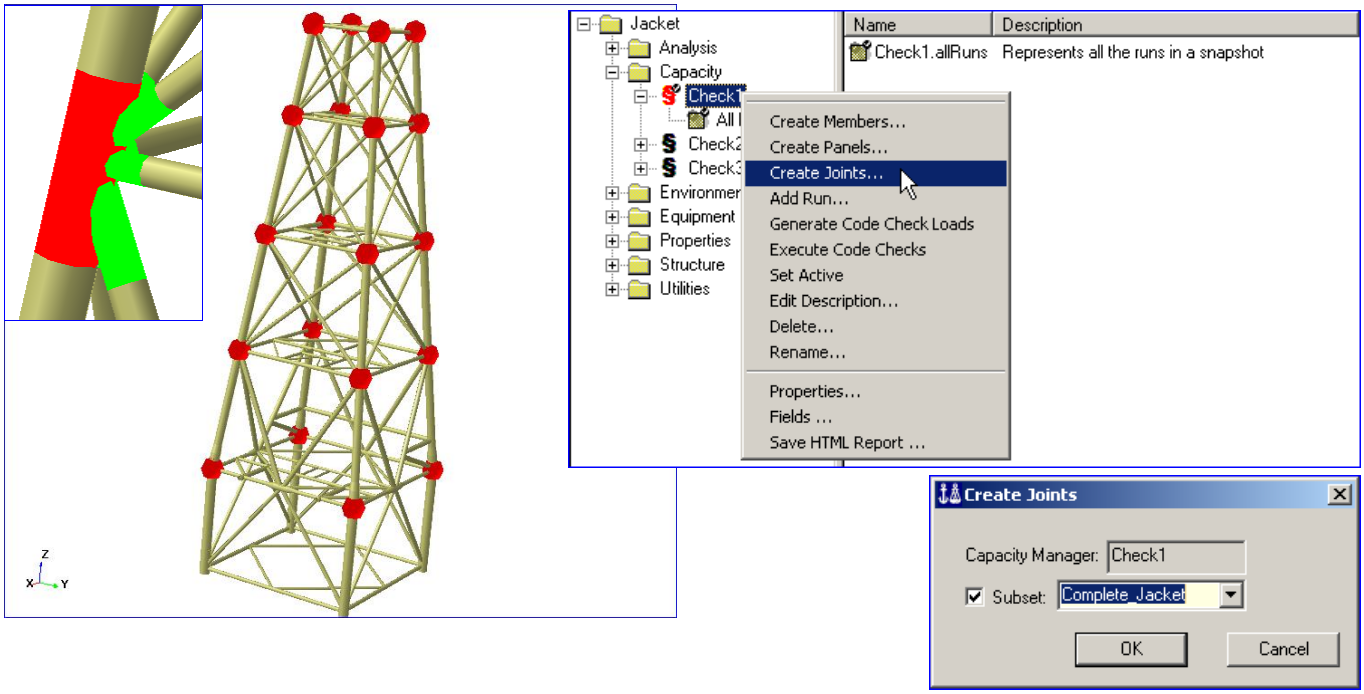


The example below illustrates the difference between the various options of splitting the beams into capacity members.

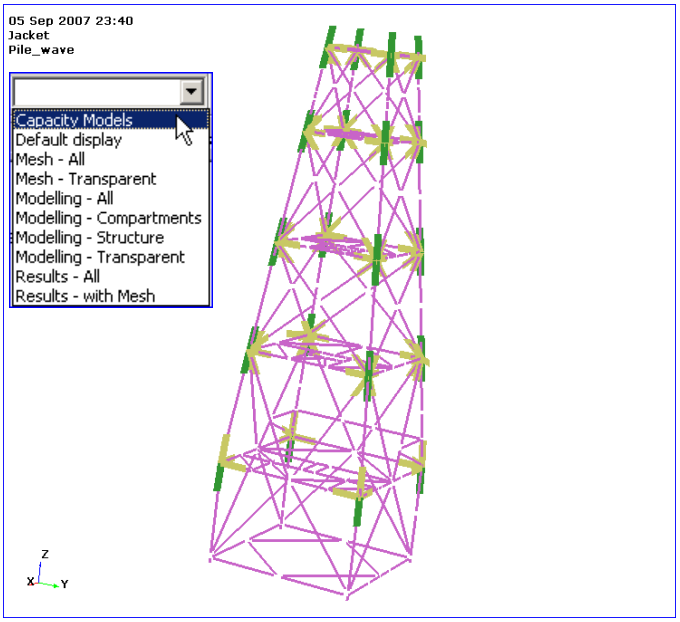


In order to make joints subjected to punching shear check, the concept model must contain joint definitions. The effects of reinforcement types like can and stubs are also accounted for. If plane-wise gap calculations have been performed, the gap values are per default used in the punching shear check.

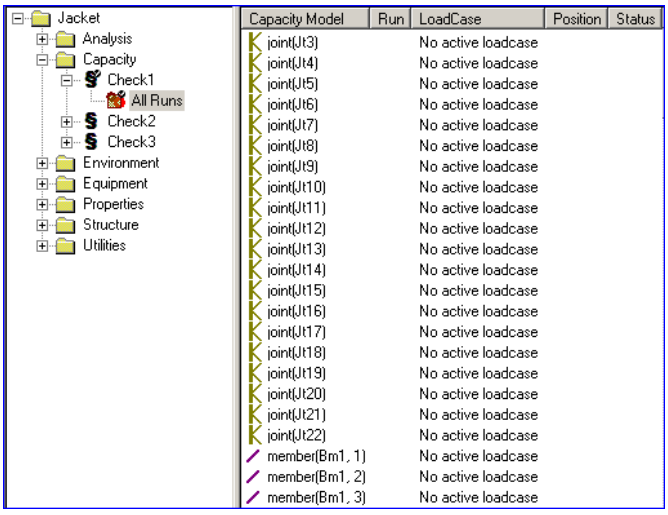
The example below shows the jacket part with a total of 20 joints – each of them has cans and stubs assigned and a plane-wise gap calculation has been performed.



The capacity model has both members and joints as shown in the picture to the right – remember to use the view called *Capacity Models*.



The browser lists both joints and members as seen below.

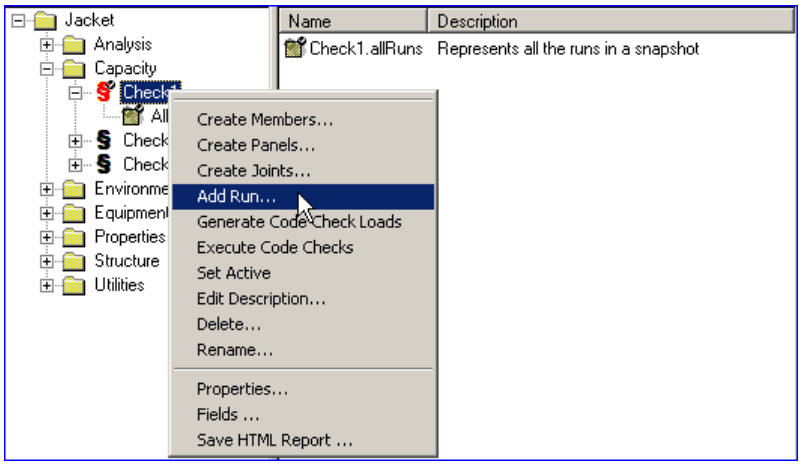


3.3 Create a code check run

During the definition of code check runs you decide

- which code of practice to use
- which loadcases to use
- global code checking parameters (i.e. those who apply to the entire capacity model) – for example buckling lengths or safety factors

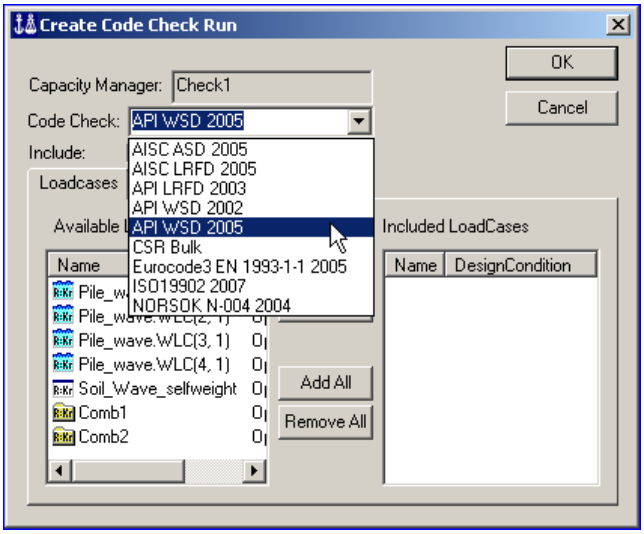
The code check run is defined from the browser.

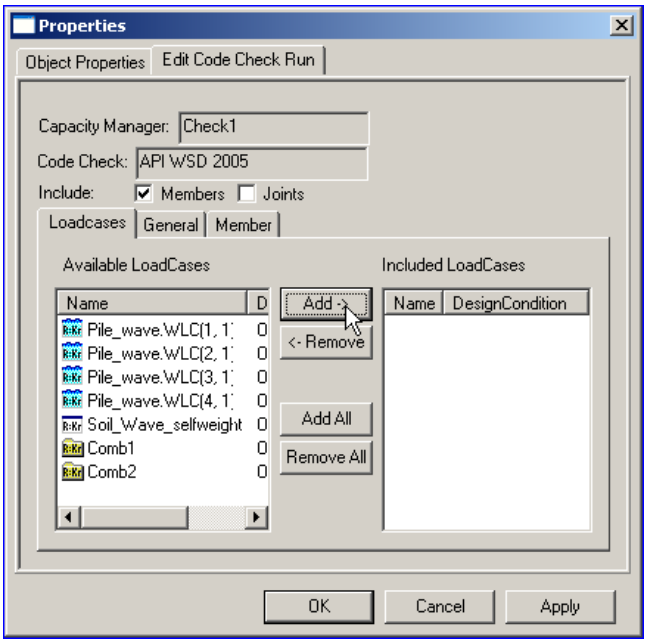


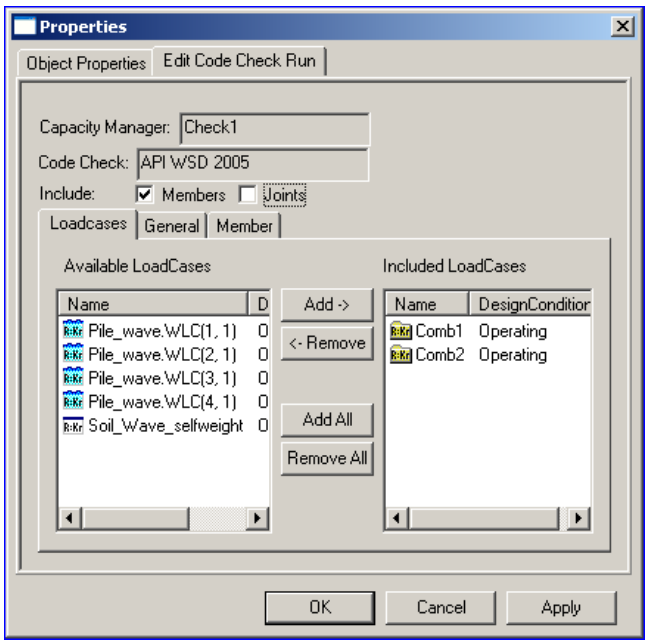
In the example to the right the code check API WSD 2005 has been selected.

This code check will not include joints; hence this option is deselected.

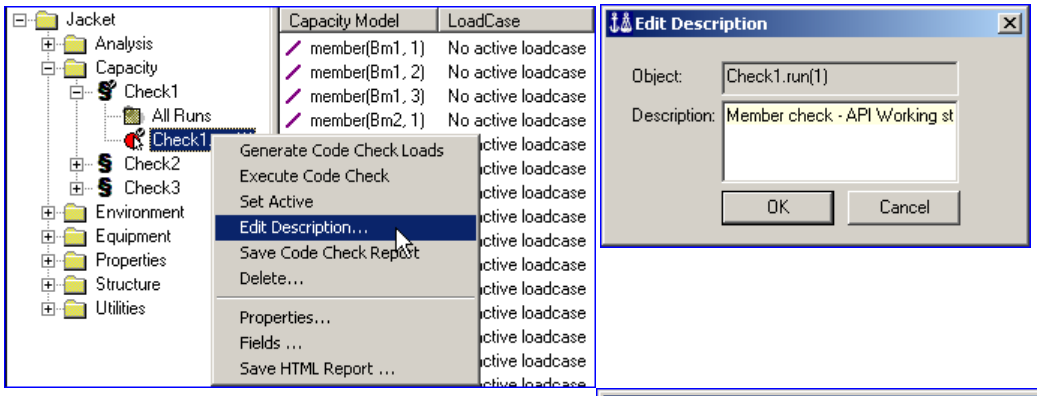
Furthermore, the two loadcases Comb1 and Comb2 have been added to the code check run.



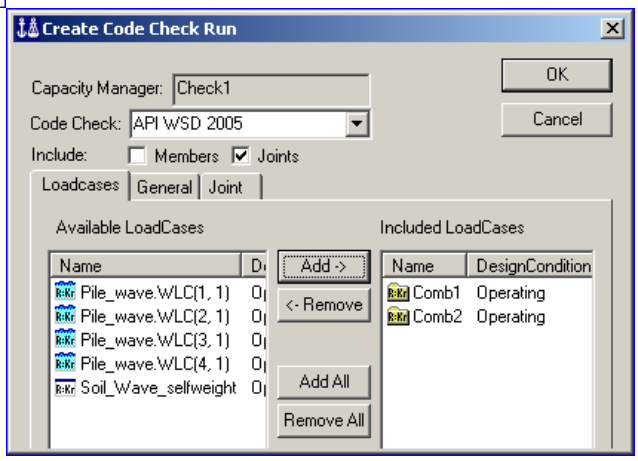


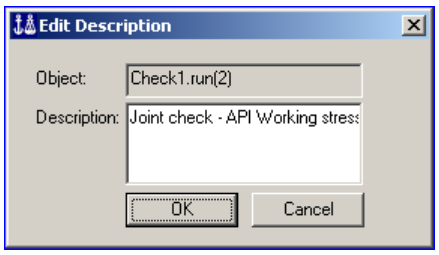


You may modify the code check run from the browser and define additional information to the run.

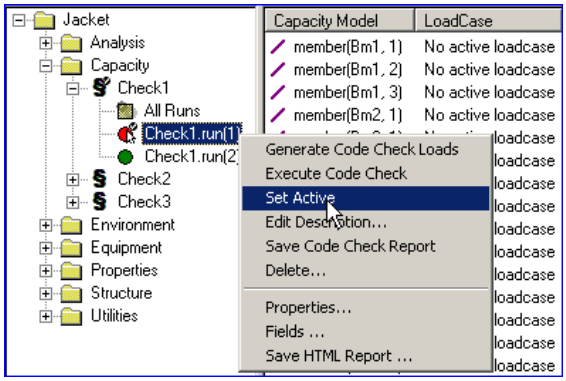


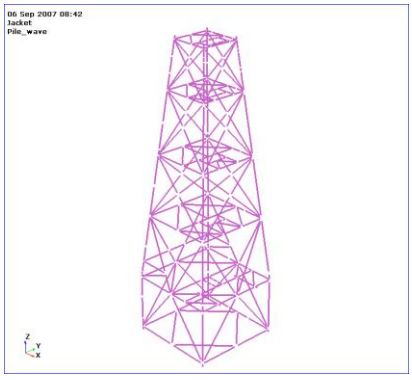
The same procedure may be used to create another code check run, typically for joints only. The only difference is that you deselect members as shown on the picture and specify another description to the run.

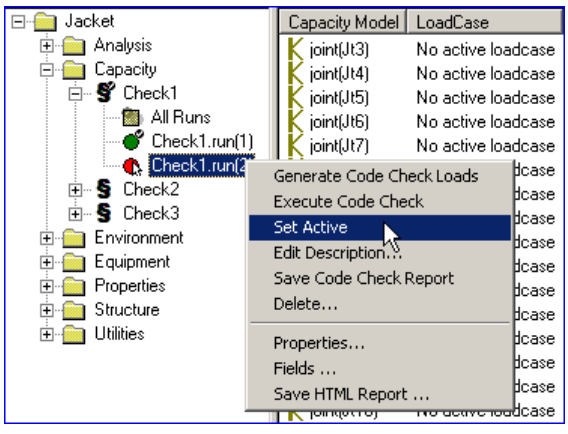


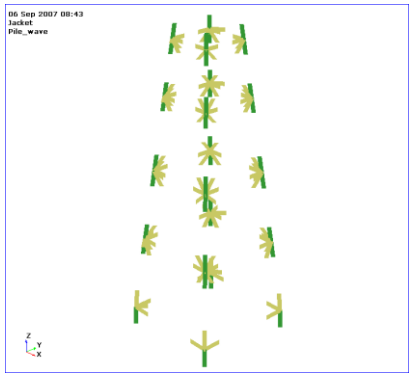


There are now two code check runs (Check1.run(1) and Check1.run(2)) for the capacity manager “Check1”. You specify which one is active from the browser:



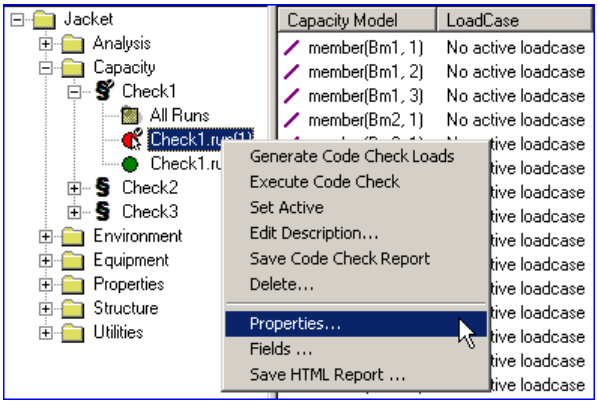






3.3.1 Define global general code checking parameters

You specify and modify the general code checking parameters when you define the code check run, or you may modify at a later stage. To modify, select Properties as shown on the picture to the right.

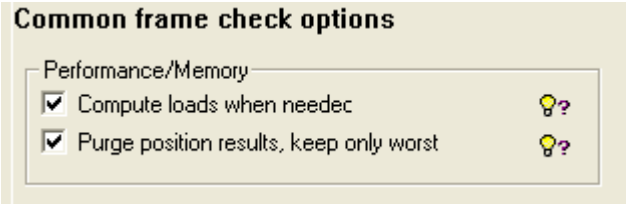


For a further description of parameters shown herein, please consult the relevant codes of practices (see Appendix B for references).

The general code checking parameters are global, i.e. they apply to all capacity members and joints.

The program default settings are shown in dialogues.

Some options regarding performance and memory usage can be found below the general tab in every frame code check. These are explained below.



Compute loads when needed

- To reduce use of database memory, you can compute temporary loads (during codecheck execution). These loads will be deleted immediately when no longer needed.
- This option can affect performance on redesign, as loads must be recalculated locally every time you change member/joint settings.
- With this option checked, you will always use the latest FEM loads. When unchecked, you will use the FEM loads retrieved the last time you used “Generate Code Check Loads”.
- Note that with option checked member loads will not be available in the report nor in object properties.

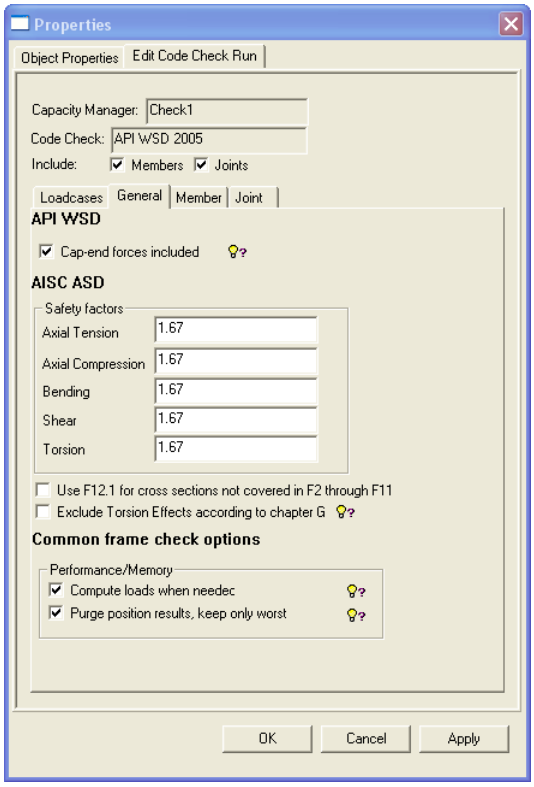
Purge position results, keep only worst

- Only worst result along a beam will be kept.
- This option reduces use of database memory.
- Note that with option checked results for other positions than the worst one will not be available in the report nor in object properties.

3.3.1.1 General parameters API WSD 2002&2005

The general parameters for the offshore code check API WSD 2002 as well as for API WSD 2005 are shown to the right.

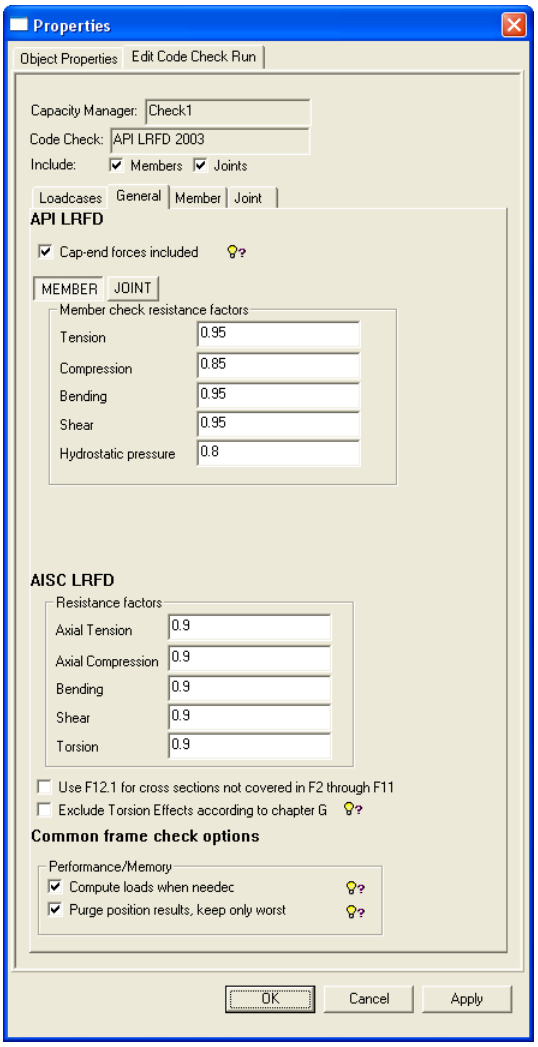
These code checks include AISC ASD 2005 for non-tubular members.

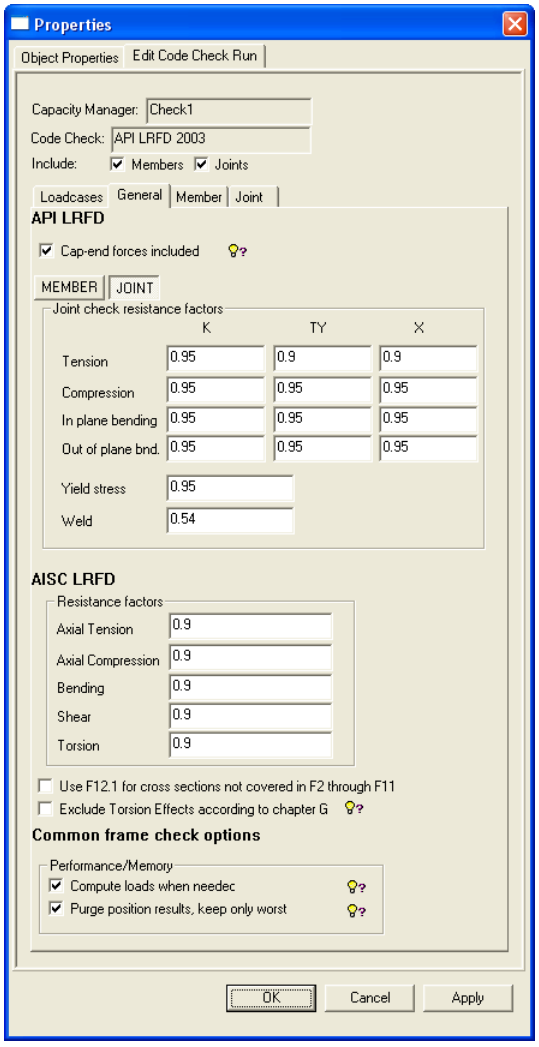


3.3.1.2 General parameters API LRFD 2003

The general *member* parameters for the offshore code check API LRFD 2003 are shown to the right.

This code check include AISC LRFD 2005 for non-tubular members.



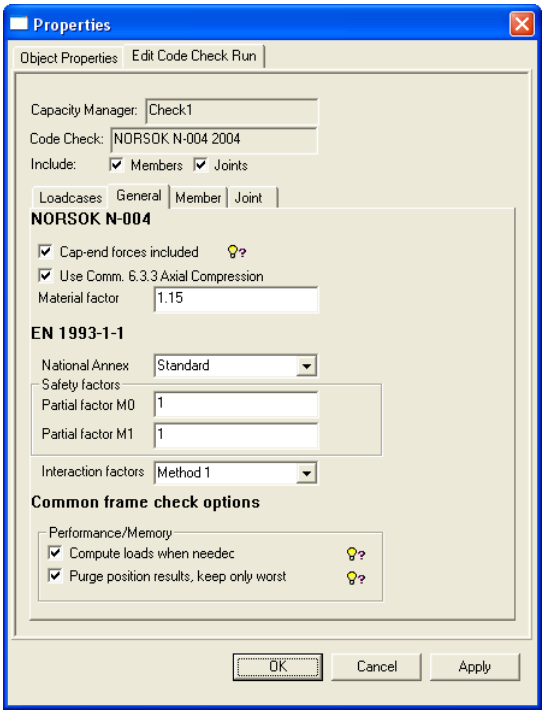


The general *joint* parameters for the offshore code check API LRFD 2003 are shown to the left.

3.3.1.3 General parameters NORSOK N004

The general parameters for the offshore code check NORSOK N004 are shown to the right.

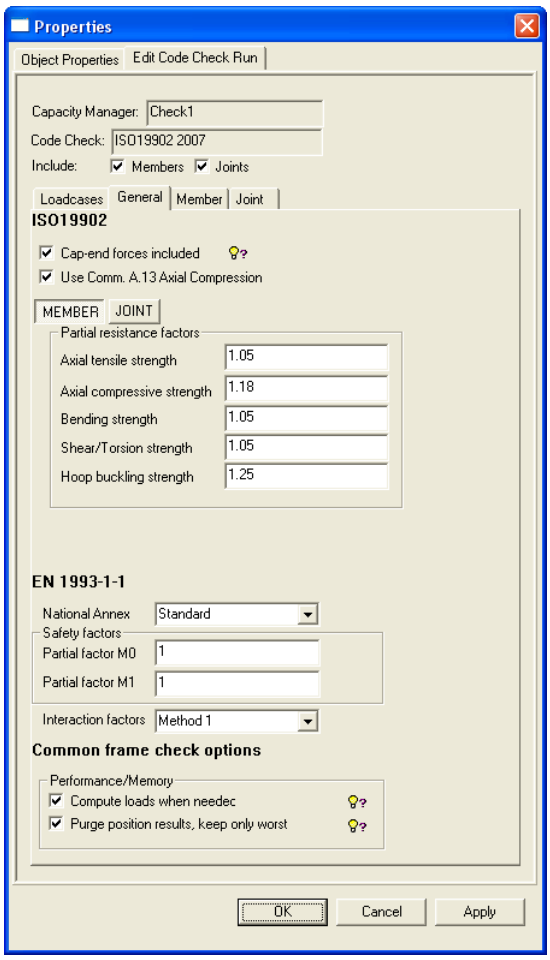
This code check includes EUROCODE 3 EN 1993-1-1 2005 for non-tubular members.



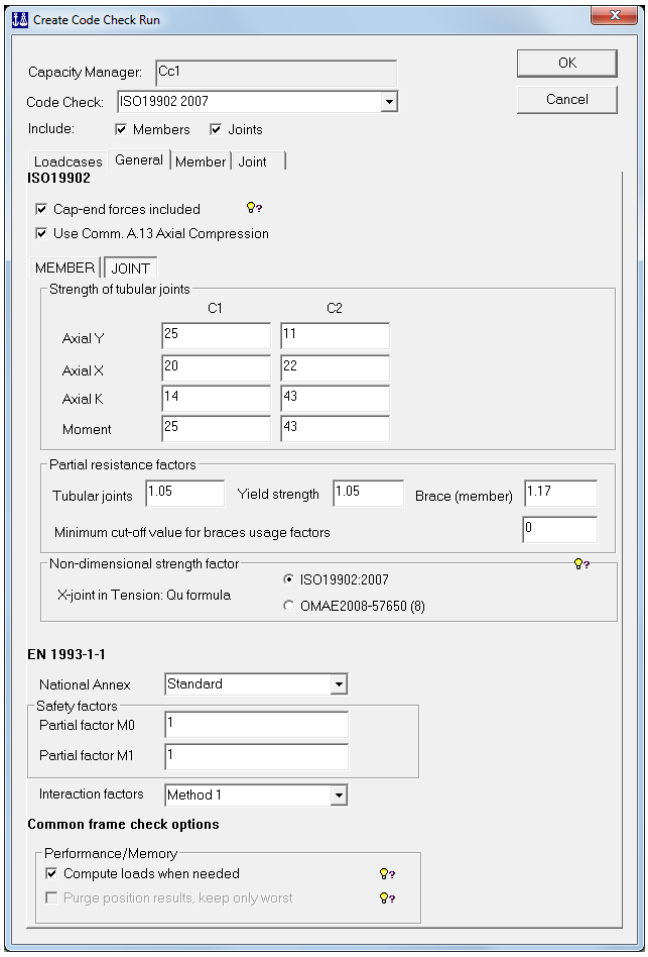
3.3.1.4 General parameters ISO 19902 2007

The general *member* parameters for the offshore code check ISO 19902 2007 are shown to the right.

This code check includes EUROCODE 3 EN 1993-1-1 2005 for non-tubular members.

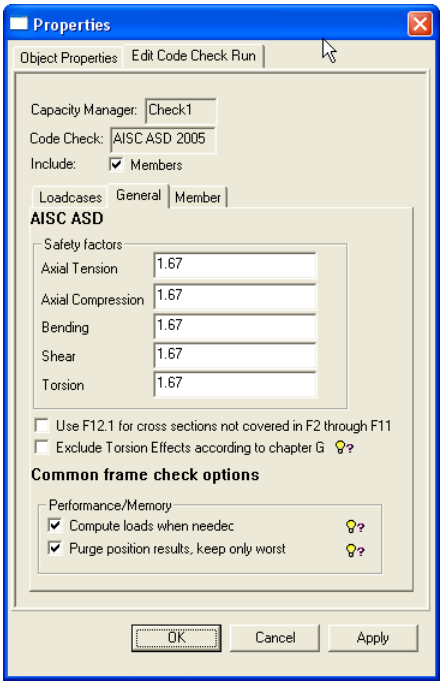


The general *joint* parameters for the offshore code check ISO 19902 2007 are shown to the right



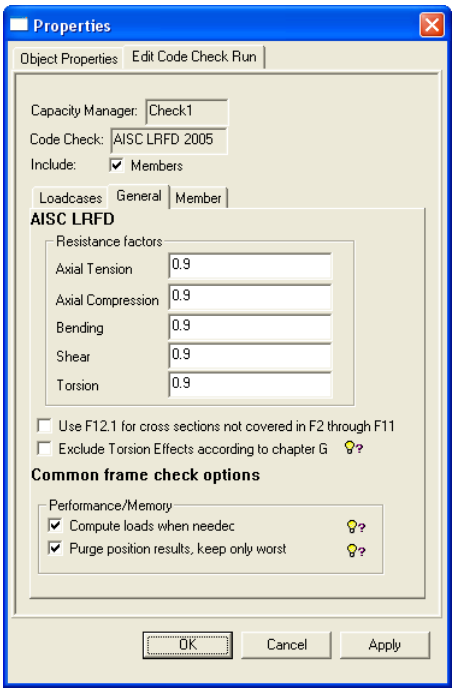
3.3.1.5 General parameters AISC ASD 2005

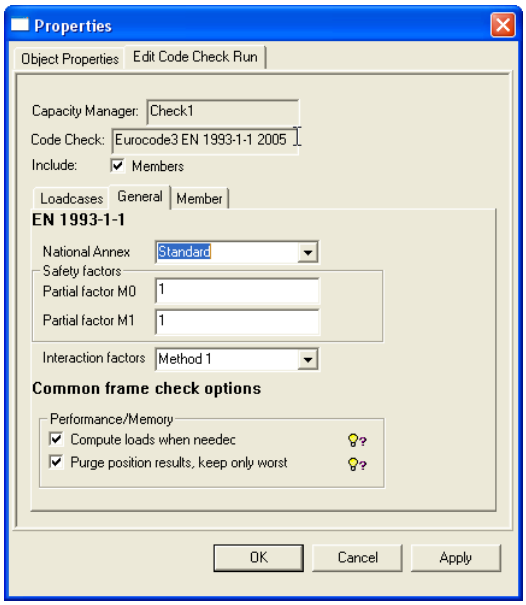
The general parameters for the onshore code check AISC ASD 2005 are shown to the right.



3.3.1.6 General parameters AISC LRFD 2005

The general parameters for the onshore code check AISC LRFD 2005 are shown to the right.

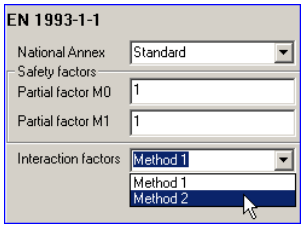


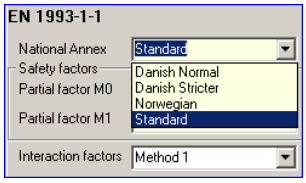


3.3.1.7 General parameters EUROCODE 3 EN 1993-1-1

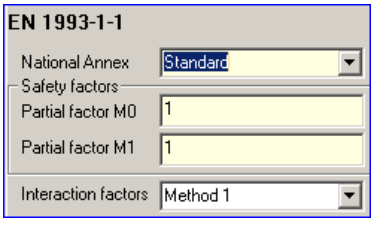
The general parameters for the onshore code check EUROCODE 3 EN 1993-1-1 2005 are shown to the left.

The national annexes may be chosen from Standard, Norwegian, Danish Normal and Danish Stricter while Interaction factors vary from Method1 and Method2.

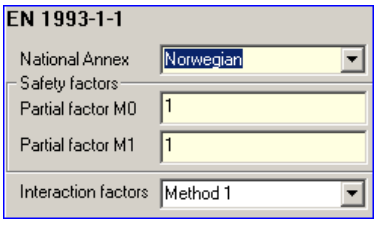




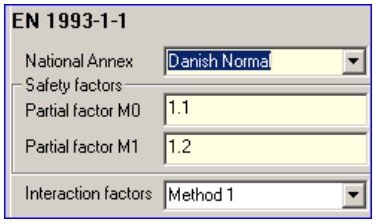
The *Standard* annex uses factors M0 and M1:



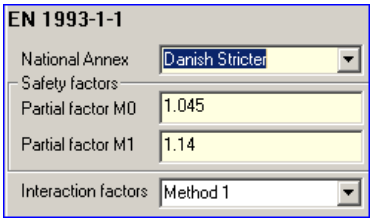
The *Norwegian* annex uses factors M0 and M1:



The *Danish* annex *Control Extent Normal* uses factors M0 and M1:

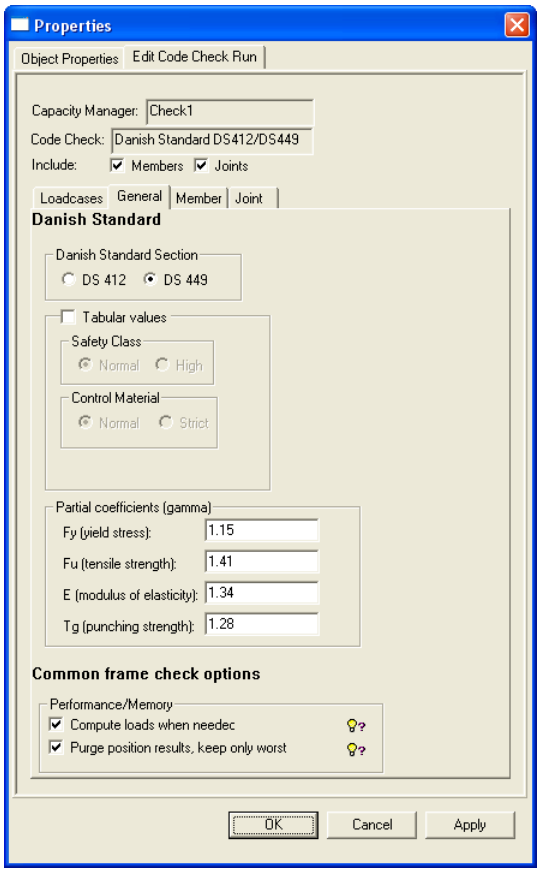


The *Danish* annex *Control Extent Stricter* uses factors M0 and M1:



3.3.1.8 General parameters DANISH STANDARD 412 / 449

The general parameters for the Danish Standard (DS) 412/449 are shown to the right. DS412 and DS449 are both made for handling tubular sections only.



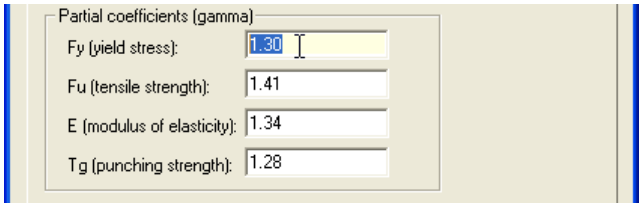
Danish Standard Section:

- DS 412 – Onshore structures
- DS 449 – Offshore structures

Safety Class

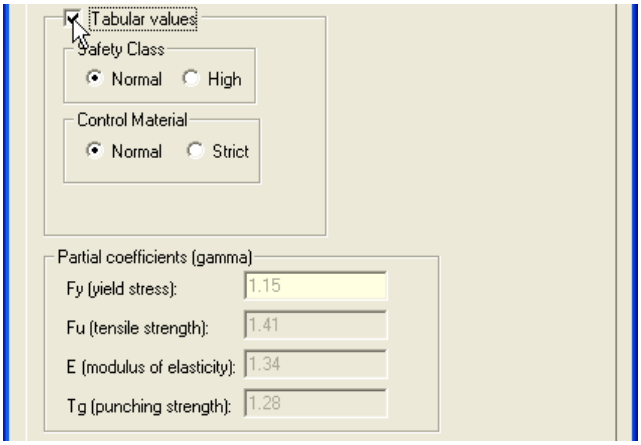
The default DS partial coefficients can be seen at the bottom of the dialog. Note that these are not the same for DS 412 and DS 449. We will first have a look at the settings for DS 449.

You can change the default partial coefficients. An example is shown in the illustration to the right

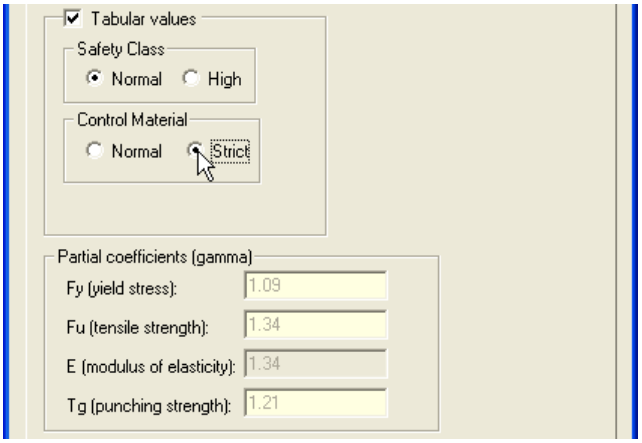


If you check off "Tabular values", the choices under "Safety Class" and "Control Material" decide which partial coefficients that will be used.

Note that the setting we just changed (Fy) is back at the value decided by the tabular values settings.



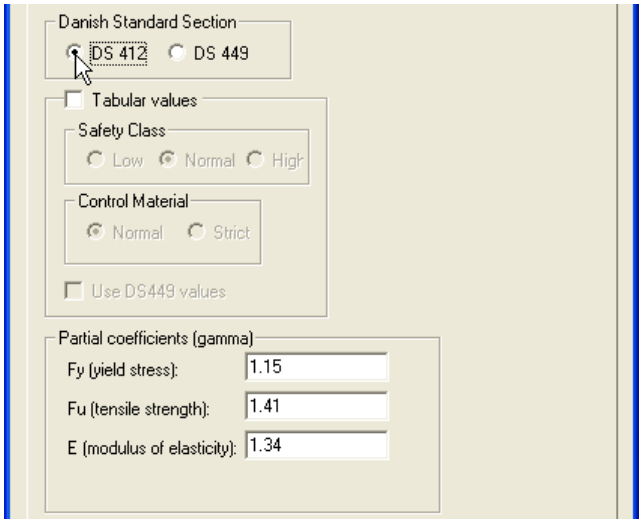
If we, as an example, change the control material from “Normal” to “Strict” by clicking the radio button, the partial coefficients change according to the new tabular values settings.



The settings for DS 412 are similar to the settings for DS 449, but there are a couple of differences:

DS 412 has a “Low” safety class.

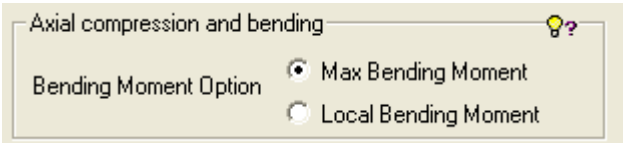
There are no Tg (punching strength) for DS 412 since this standard do not support joints.



3.3.2 Define global member parameters

The global member parameters (those who apply to the all capacity members) may be changed from default values when you define the code check run or later. The default values are shown in the following for each code check alternative.

Some options regarding axial compression and bending can be found below the member tab in all the frame code checks except Danish standard and Eurocode.
These are explained below.



Axial compression and bending - Bending Moment Option

Max Bending Moment

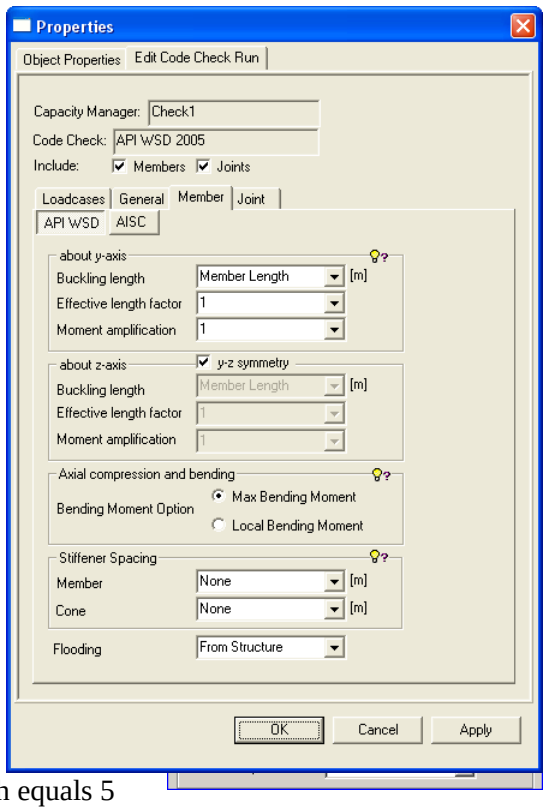
This option selects the maximum bending moments along a capacity member derived by the effect of moment gradient, Cm. This method is considered to be best practise.

Local Bending Moment

This option uses the local bending moments at every code check positions.

3.3.2.1 Member parameters API WSD 2002 & 2005

The default member data for *tubular* members are shown to the right.



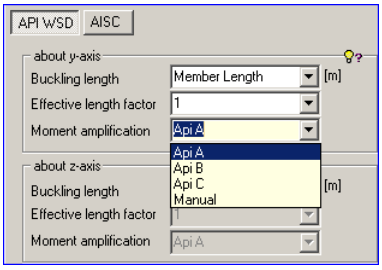
Notice that there are different properties for tubular members (API WSD) and non-tubular members (AISC). GeniE will automatically detect which profiles are present in the capacity model.

For *non-tubular* members, the AISC ASD 2005 is used.

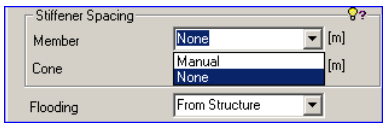
Each parameter for API WSD 2002 & 2005 is explained in the following.

- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 5 m times 0.8 = 4 m.

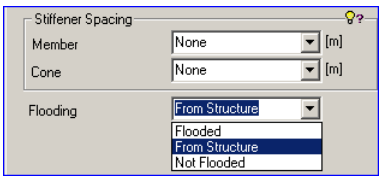
- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. For tubular members the automatic calculations may be in accordance with API formulas A, B or C. You may also specify the moment amplification factor manually.



- The spacing between ring stiffeners for tubular and conical members may be specified. The default value corresponds to the member length, i.e. it is assumed that no internal ring stiffeners are present. This value is used for hydrostatic collapse and stability calculations (for tubular members only).

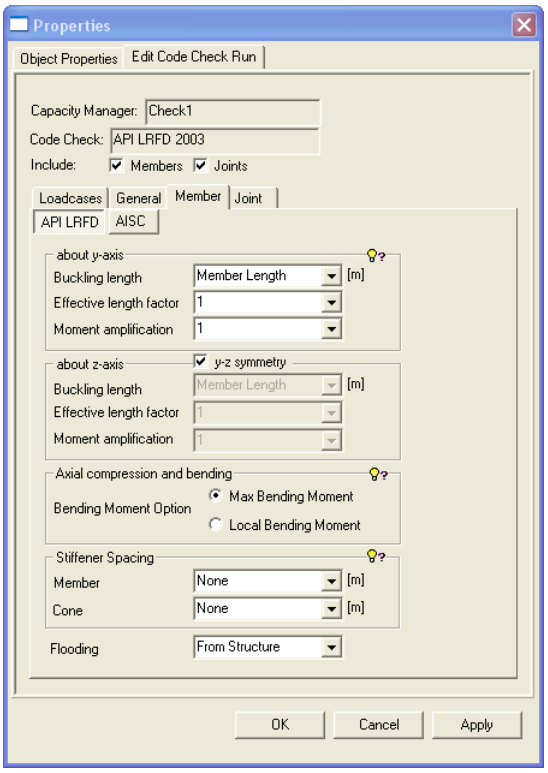


- The flooding status (either non-flooded, or flooded) is used when performing a member check of a member with tubular cross section exposed to hydrostatic water pressure. The flooding status may be the same as defined in the concept model or it may be given manually. The default value is the same as in the concept model.



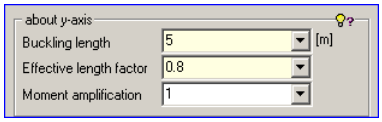
3.3.2.2 Member parameters API LRFD 2003

The default member data for tubular members are shown to the right.



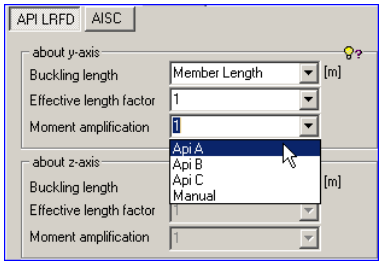
Notice that there are different properties for tubular members (API WSD) and non-tubular members (AISC). GeniE will automatically detect which profiles are present in the capacity model.

For *non-tubular* members, the AISC LRFD 2005 is used.

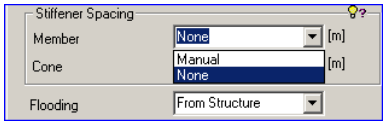


Each parameter for API LRFD is explained in the following.

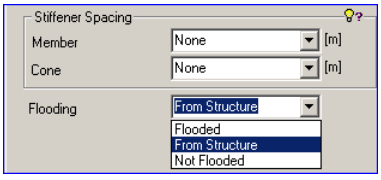
- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 5 m times 0.8 = 4 m.



- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. For tubular members the automatic calculations may be in accordance with API formulas A, B or C. You may also specify the moment amplification factor manually.

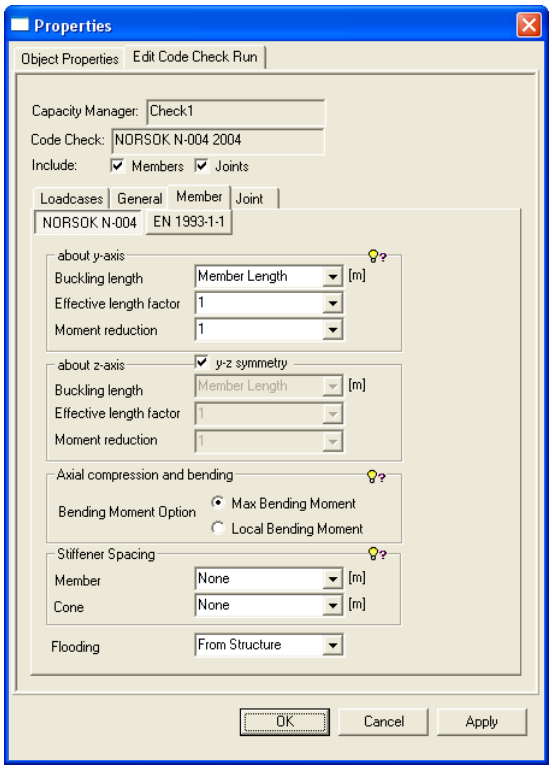


- The spacing between ring stiffeners for tubular and conical members may be specified. The default value corresponds to the member length, i.e. it is assumed that no internal ring stiffeners are present. This value is used for hydrostatic collapse and stability calculations (for tubular members only).



- The flooding status (either non-flooded, or flooded) is used when performing a member check of a member with tubular cross section exposed to hydrostatic water pressure. The flooding status may be the same as defined in the concept model or it may be given manually. The default value is the same as in the concept model.

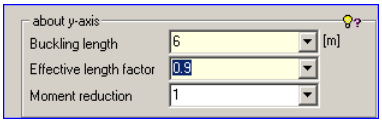
3.3.2.3 Member parameters NORSOK N004



The default member data for tubular members are shown to the right.

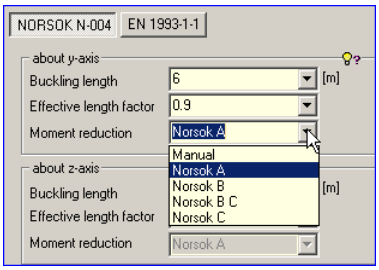
Notice that there are different properties for tubular members (NORSOK) and non-tubular members (EUROCODE). GeniE automatically detects which profiles are present in the capacity model and configures the input dialog accordingly.

For *non-tubular* members, the EUROCODE 3 EN 1993-1-1 is used.

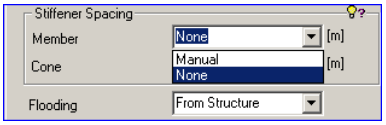


Each parameter for NORSOK is explained in the following.

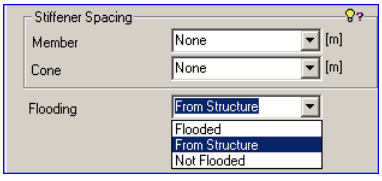
- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 6 m times 0.9 = 5.4 m.



- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. For tubular members the automatic calculations may be in accordance with NORSOK formulas A, B, B C or C. You may also specify the moment amplification factor manually.

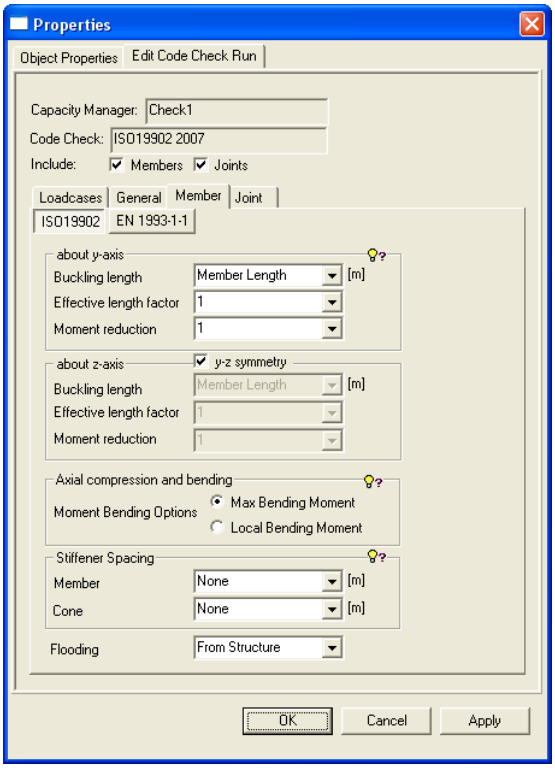


- The spacing between ring stiffeners for tubular members may be specified. The default value corresponds to the member length, i.e. it is assumed that no internal ring stiffeners are present. This value is used for hydrostatic collapse and stability calculations (for tubular members only).
- Stiffener spacing for cones- same as stiffener spacing for tubular members (see above)



- The flooding status (either non-flooded, or flooded) is used when performing a member check of a member with tubular cross section exposed to hydrostatic water pressure. The flooding status may be the same as defined in the concept model or it may be given manually. The default value is the same as in the concept model.

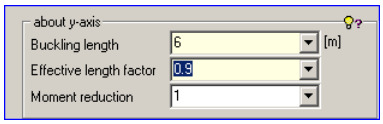
3.3.2.4 Member parameters ISO 19902 2007



The default member data for tubular members are shown to the right.

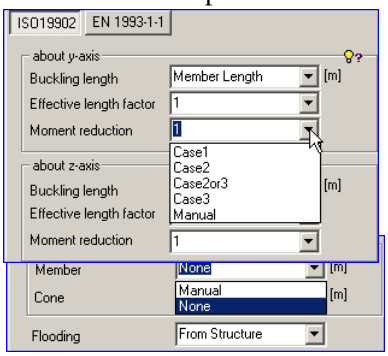
Notice that there are different properties for tubular members (NORSOK) and non-tubular members (EUROCODE). GeniE automatically detects which profiles are present in the capacity model and configures the input dialog accordingly.

For *non-tubular* members, the EUROCODE 3 EN 1993-1-1 is used.

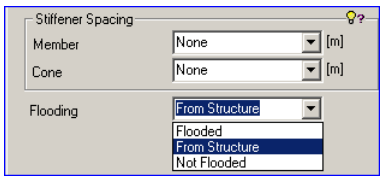


Each parameter for ISO 19902 is explained in the following.

- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 6 m times 0.9 = 5.4 m.
- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. For tubular members the automatic calculations may be in accordance with ISO formulas Case1, Case2, Case2or3 orCase3. You may also specify the moment amplification factor manually.

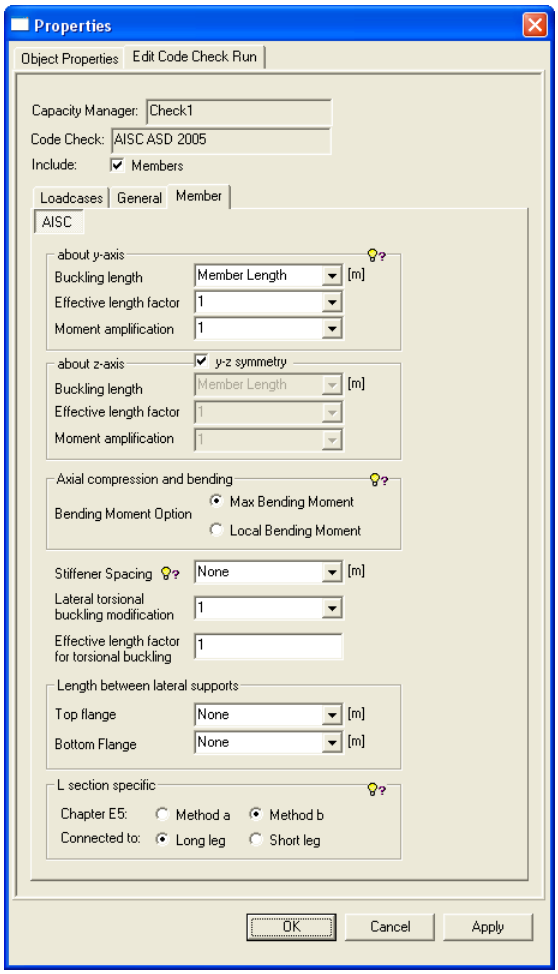


- The spacing between ring stiffeners for tubular members may be specified. The default value corresponds to the member length, i.e. it is assumed that no internal ring stiffeners are present. This value is used for hydrostatic collapse and stability calculations (for tubular members only).
- Stiffener spacing for cones- same as stiffener spacing for tubular members (see above)
- The flooding status (either non-flooded, or flooded) is used when performing a member check of a member with tubular cross section exposed to hydrostatic water pressure. The flooding status may be the same as defined in the concept model or it may be given manually. The default value is the same as in the concept model.



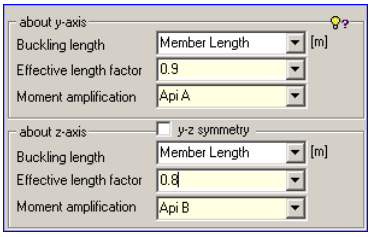
3.3.2.5 Member parameters AISC ASD 2005

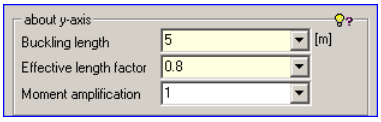
The member parameters for the onshore code check AISC ASD 2005 are shown to the right



This code check is used by API WSD 2002 & 2005 for non-tubular profiles.

Observe that it is possible to have different values for buckling lengths about y-axis and z-axis (in-plane and out-of-plane).

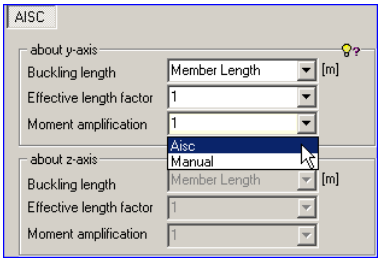




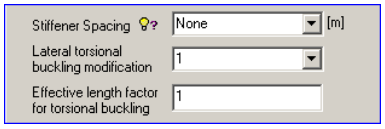
Each parameter for AISC ASD 2005 is explained in the following.

- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 5 m times 0.8 = 4 m.

- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. There is one automatic option as described in the AISC code. You may also specify the moment amplification factor manually.



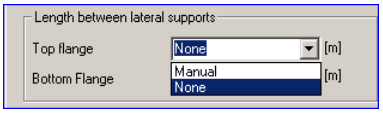
- The spacing between stiffeners for non-tubular members may be specified. The default value corresponds to the member length, i.e. it is assumed that no stiffeners are present. This value is used for computation of the shear capacity.



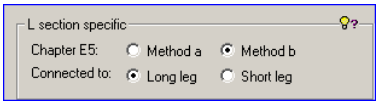
- Lateral torsional buckling modification factor for non-uniform moment diagrams, Cb (ref. AISC chapter F1.).
  The default value is 1.

- Effective length factor for torsional buckling, Kz
  (ref. AISC chapter E4.). The default value is 1.

- The length between lateral supports is used when checking the member for lateral buckling. Values may be given for both top and bottom flanges. The default value is the member length; however you may specify a specific value.



- Specific parameters used in connection with single angle compression elements, ref. chapter E5 item (a) versus (b) and how the member is connected, i.e. through long or short leg. The default values are shown to the right.

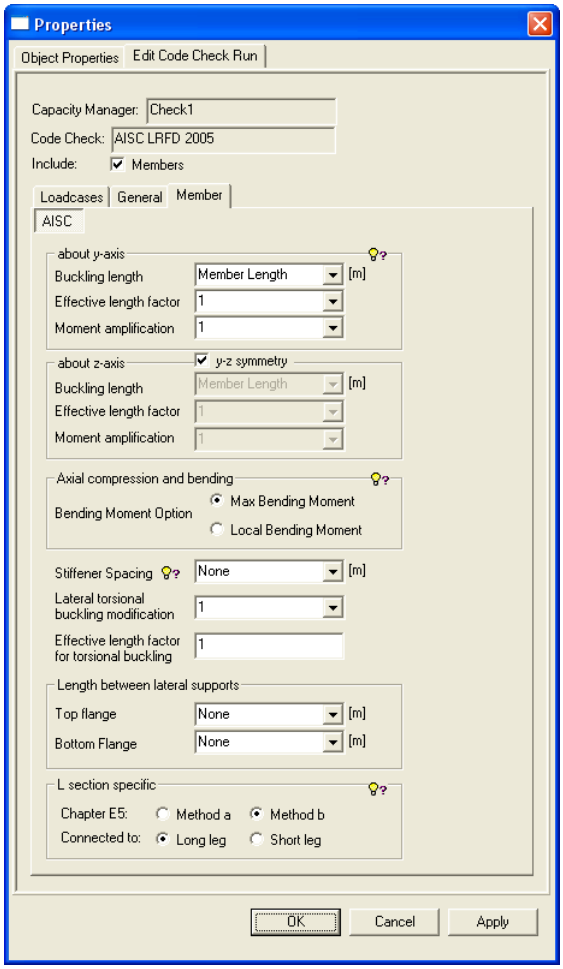


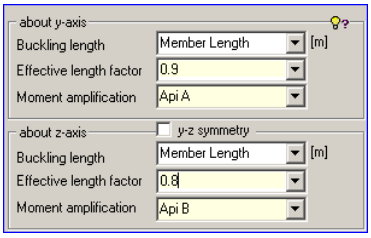
3.3.2.6 Member parameters AISC LRFD 2005

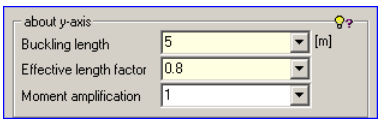
The member parameters for the onshore code check AISC LRFD 2005 are shown to the right

This code check is used by API LRFD 2003 for non-tubular profiles

Observe that it is possible to have different values for buckling lengths about y-axis and z-axis (in-plane and out-of-plane).

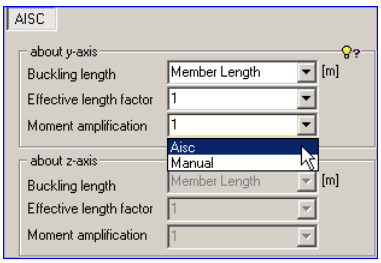




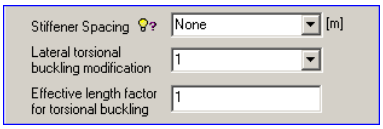


Each parameter for AISC LRFD 2005 is explained in the following.

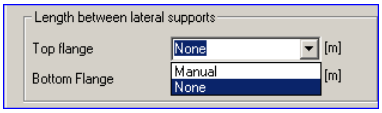
- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 5 m times 0.8 = 4 m.



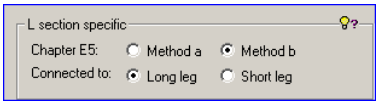
- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. There is one automatic option as described in the AISC code. You may also specify the moment amplification factor manually.



- The spacing between stiffeners for non-tubular members may be specified. The default value corresponds to the member length, i.e. it is assumed that no stiffeners are present. This value is used for computation of the shear capacity.
- Lateral torsional buckling modification factor for non-uniform moment diagrams, Cb (ref. AISC chapter F1.).
  The default value is 1.
- Effective length factor for torsional buckling, Kz
  (ref. AISC chapter E4.). The default value is 1.



- The length between lateral supports is used when checking the member for lateral buckling. Values may be given for both top and bottom flanges. The default value is the member length; however you may specify a specific value.



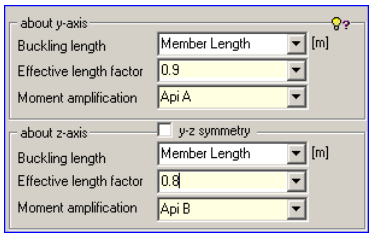
- Specific parameters used in connection with single angle compression elements, ref. chapter E5 item (a) versus (b) and how the member is connected, i.e. through long or short leg. The default values are shown to the right.

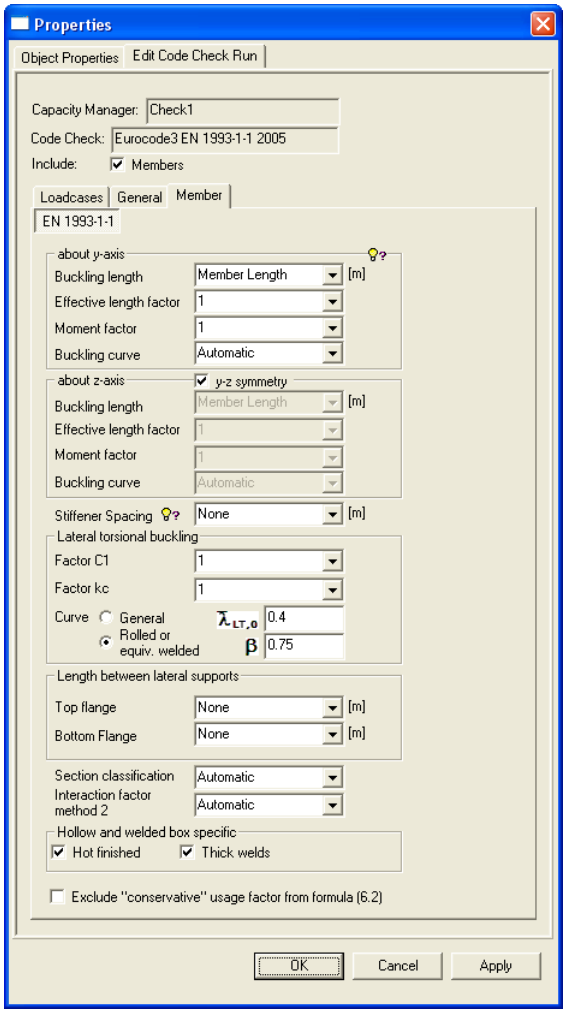
3.3.2.7 Member parameters EUROCODE 3 EN 1993-1-1

The member parameters for the onshore code check EUROCODE 3 EN 1993-1-1 are shown to the right.

This code check is used by NORSOK N004 and ISO 19902 for non-tubular profiles.

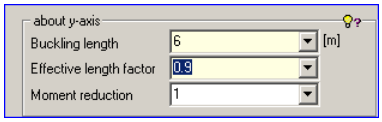
Observe that it is possible to have different values for buckling lengths about y-axis and z-axis (in-plane and out-of-plane).



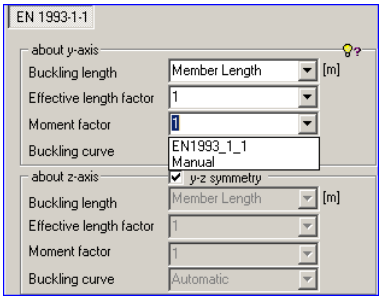


Each parameter for EUROCODE 3 EN 1993-1-1 is explained in the following.

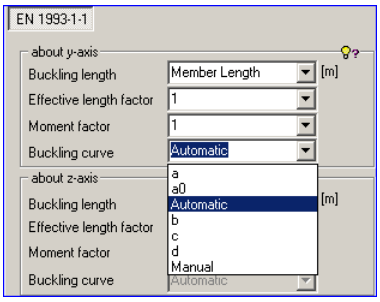
- The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined. In the example to the right, the buckling length equals 6 m times 0.9 = 5.4 m.



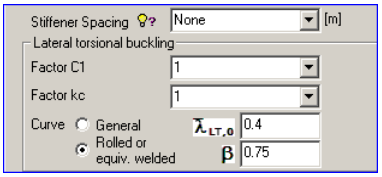
- The moment amplification allows for explicit or automatic calculation of moment amplification reduction factors, to account for secondary moments due to axial loads in buckling calculations. There is one automatic option as described in the NORSOK code. You may also specify the moment amplification factor manually.



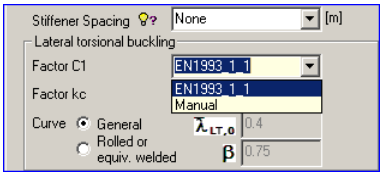
- The buckling curves may be defined automatically by the program dependent of profile type. Alternatively define which curve to be used. (Ref. EN 1993-1-1 Table 6.2) Note that automatic selection for "Hollow and welded box" sections also require information with respect to "Hot finished" and "Thick welds". The default value is automatic definition according to the profile type.



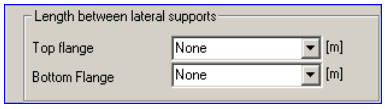
- Stiffener spacing means length between vertical stiffeners in webs. The default value is "None" meaning that there are no vertical stiffeners along the member, i.e. the stiffener spacing is the member length.



- Manually input or automatic calculation of lateral torsional buckling parameters C1 and kc in addition to curve selection/method for use in calculation of χ_{LT} . (Ref EN 1993-1-1 section 6.3.2). The default values are shown to the right.
  - An example on selection of the factor C1 is also shown.

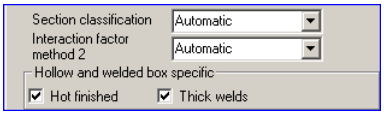


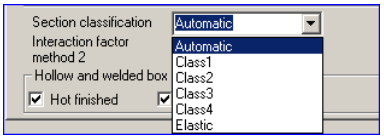
- The length between lateral supports is used when checking the member for lateral buckling. Values may be given for both top and bottom flanges. The default value is the member length (i.e. no lateral support present); however you may specify a specific value.

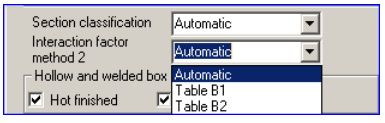


- Select how to handle the cross-section classification, i.e.
  - manually give the class
  - automatic classification
  - elastic = automatic classification, but limited to class 3 and 4

  The default values are shown to the right. There are also examples on the selection of "section classification" and "interaction factor method 2".

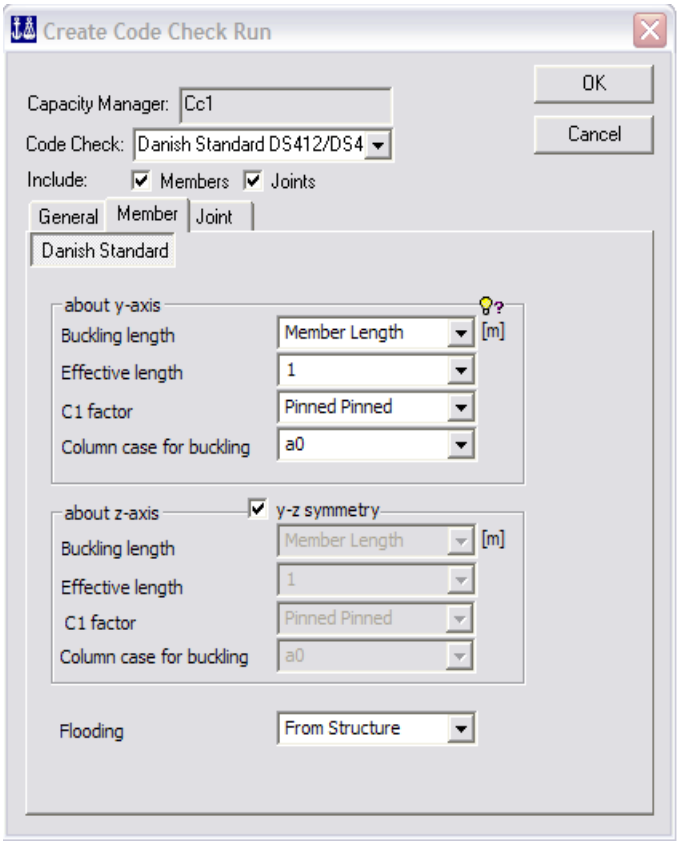






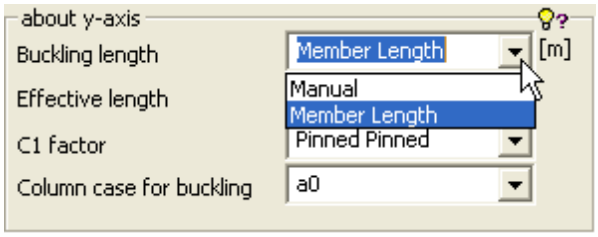
3.3.2.8 Member parameters DANISH STANDARD 412 / 449

The member parameters for the onshore code check Danish Standard 412 (DS412) and the offshore code check Danish Standard 449 (DS449) are shown to the right.



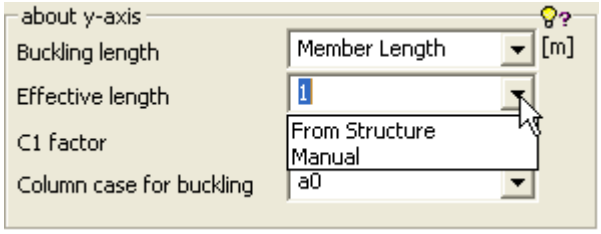
Observe that it is possible to have different values for buckling lengths, effective lengths, C1 factors and column case for buckling about y-axis and z-axis (in-plane and out-of-plane).

About y-axis



Buckling length

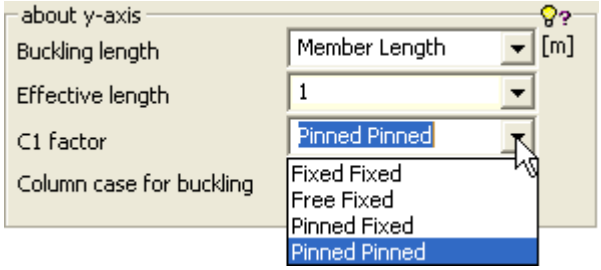
- Manual
- Member Length



Effective length

- From Structure
- Manual

The buckling length is the product of member length and effective length factor. The member length may be derived from the capacity member or explicitly defined.

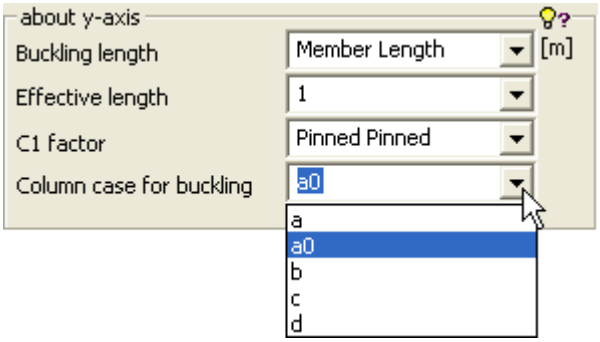


C1 factor is the parameter used to compute internal parameters for circular cylindrical shell as function of the boundary conditions:

- Fixed Fixed
- Free Fixed
- Pinned Fixed
- Pinned Pinned

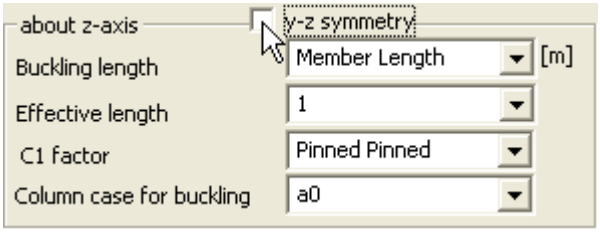
Column case for buckling

The significance of the residual stresses on the shape and manufacture of the profile is considered. You can select between 5 cases, *a0*, *a*, *b*, *c* and *d*. A description of the cases can be found in Table V 6.2.1 in the reference documents for DS



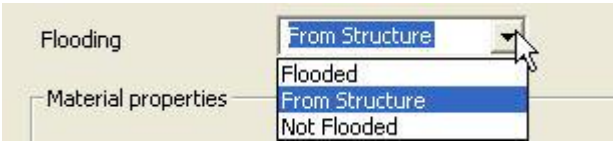
About z-axis

By unchecking the "y-z symmetry" checkbutton you can change the same settings as for the y-axis.



Flooding

- Flooded
- From Structure
- Not Flooded

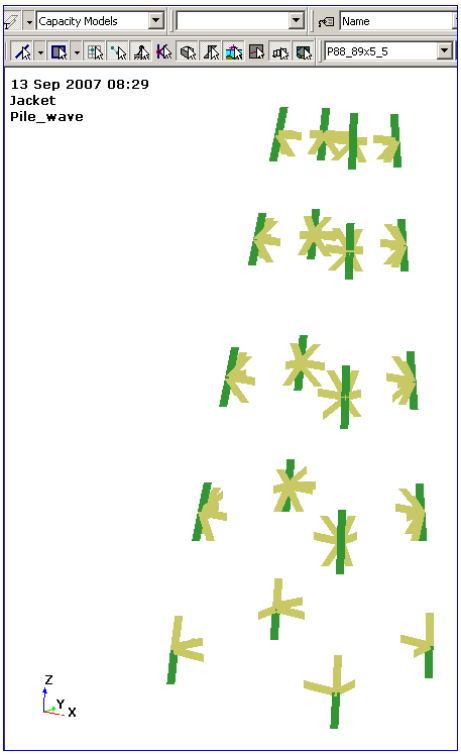


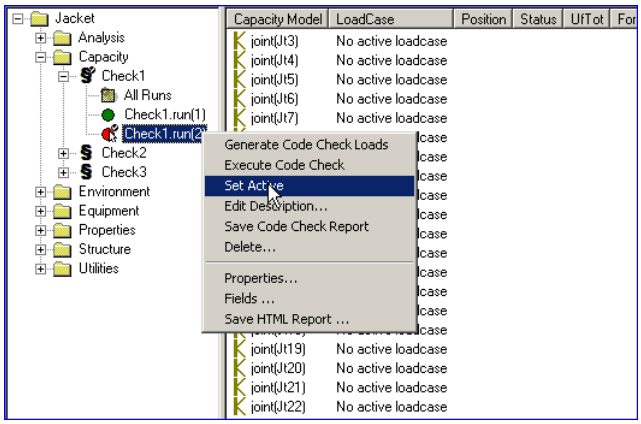
The flooding status (either non-flooded, or flooded) is used when performing a member check of a member with tubular cross section exposed to hydrostatic water pressure. The flooding status may be the same as defined in the concept model or it may be given manually. The default value is the same as in the concept model (from structure).

3.3.3 Define global joint parameters

When defining or modifying the global joint parameters make sure you either define a run where joints are present or set an existing run including joints to active.

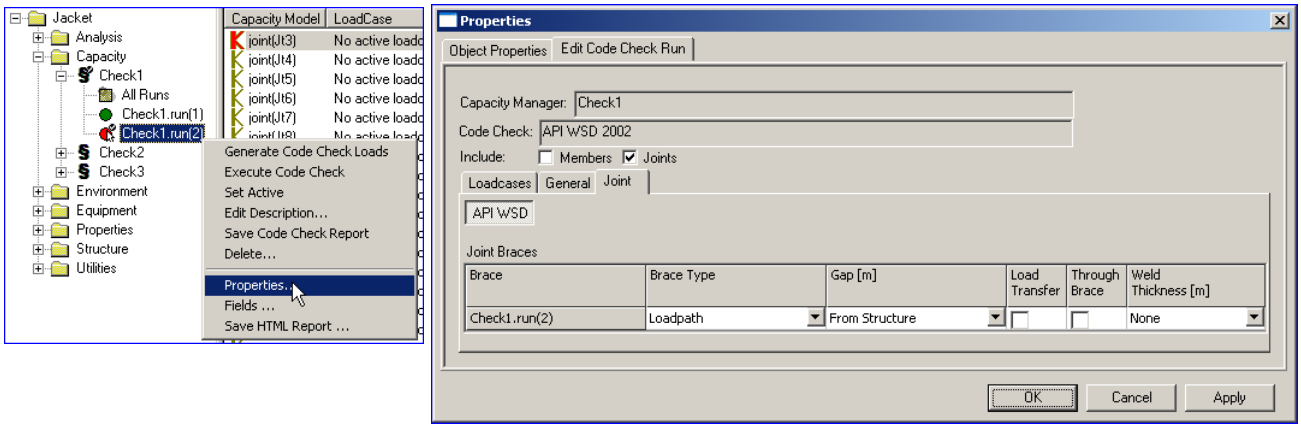
You should then see a capacity model that contains the capacity joints (remember to set the view to “Capacity Models”.





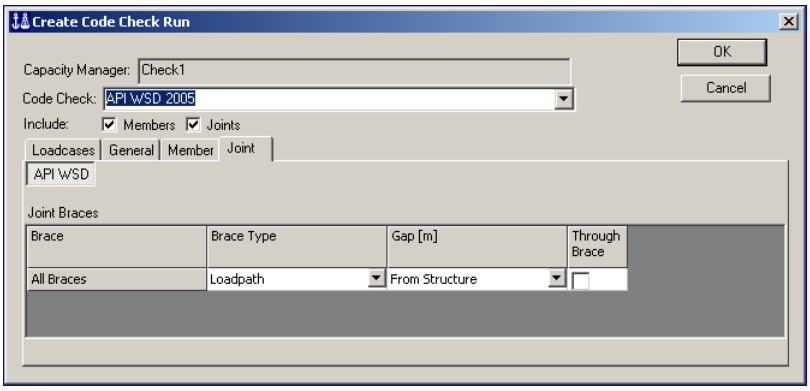
The global joint parameters are defined when you create the run or modify an existing run. In this case the latter option is shown (the input dialog form is the same when you define a run).

The global default parameters for capacity joints are shown below and they may be modified accordingly. The various parameters are described in the following.

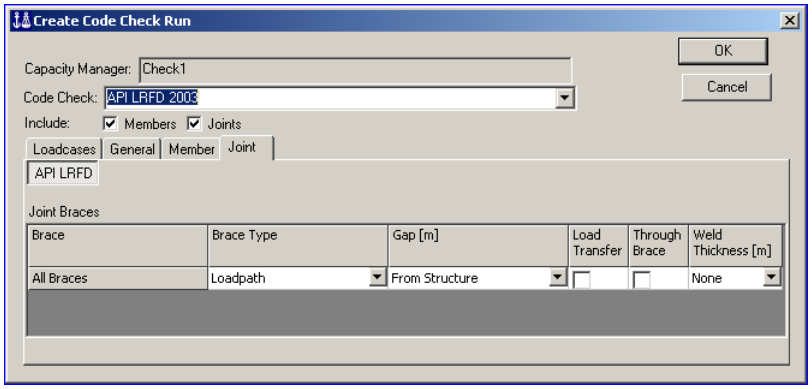


The global joint parameters for API WSD 2002 are shown above.

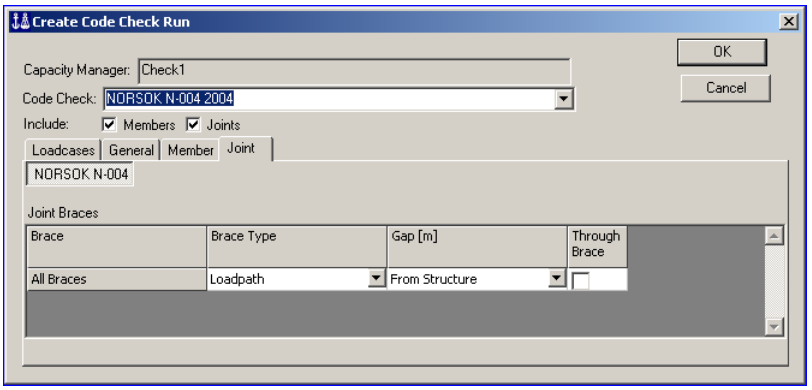
The global joint parameters for API WSD 2005 are shown to the right.



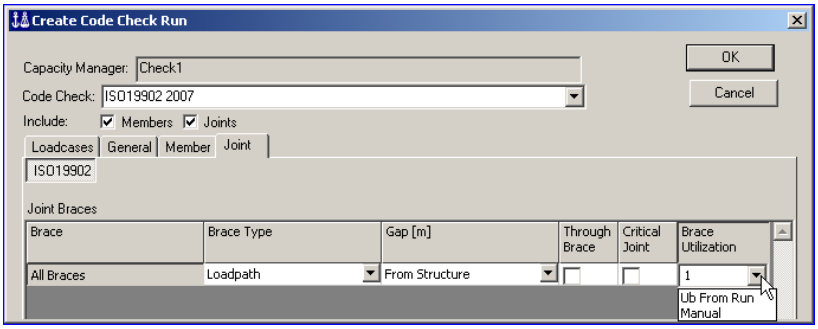
The global joint parameters for API LRFD 2003 are shown to the right.



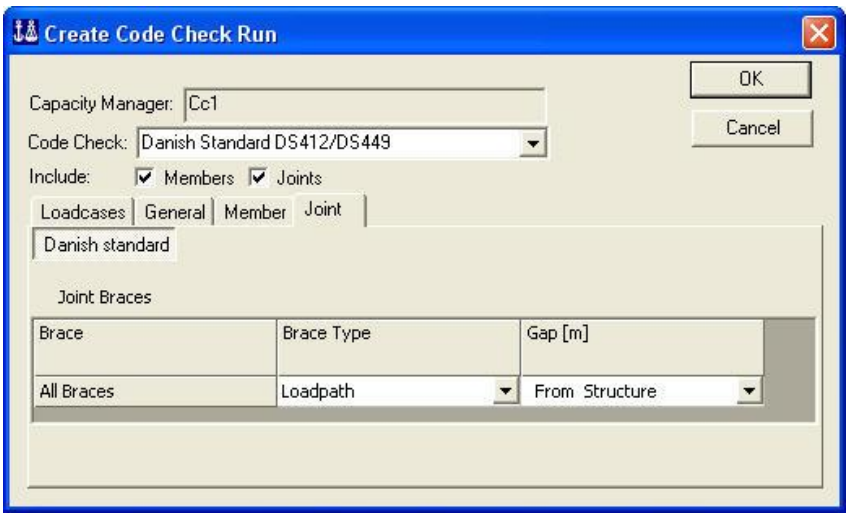
The global joint parameters for NORSOK N-004 are shown to the right.



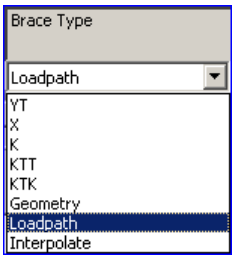
The global joint parameters for ISO 19902 are shown to the right.



The global joint parameters for DS are shown to the right



3.3.3.1 Brace type classification

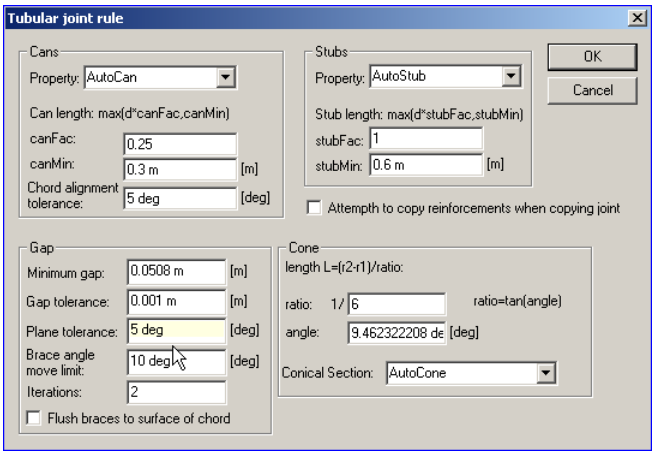


- YT (T or Y joint)
- X
- K
- KTT (T part of a KT joint)
- KTK (K part of a KT joint)
- Geometry (the choice of joint classification is based on the actual geometry)
- Loadpath (the choice of joint classification is based on the actual geometry and then force distribution) - by default, all joints are assumed to be Loadpath
- Interpolate

According to the code checking standards it is recommended to the classification based on load path.

GeniE will count the number of braces in the same plane as the current brace and the chord element, and based on the number of near and far side braces determine the joint type. The plane as specified in the rules for tubular joint design (from menu *Edit*|*Rules*|*Joint Design*) is used to determine if neighbouring braces are in the same plane as the current brace.

The example to the right shows a plane angle of 5 degrees.

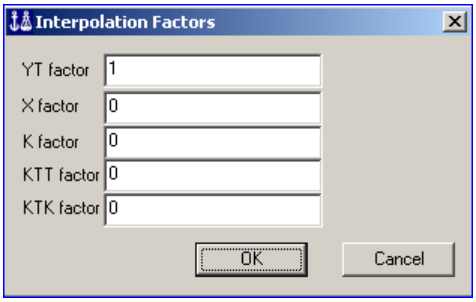


The joint classification ('Geometry' or 'Loadpath') is done as follows.

Number of braces		Joint type based on		
Same side	Opposite side	Geometry (only)	Loadpath (possibly in addition)	
1	0	YT		
2	0	K	YT	
3	0	KTK	YT	(upper/lower brace)
3	0	KTT	YT	(middle brace)
4	0	Impossible		
1	>0	X	YT	
2	>0	K	YT, X	
3	>0	KTK	YT, X	(upper/lower brace)
3	>0	KTT	YT, X	(middle brace)
4	>0	Impossible		

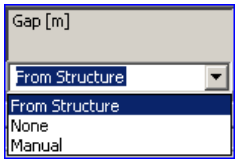
Please notice that when using 'Loadpath' the joint classification may vary from load case to load case.

When using the interpolate option, you may interpolate between YT, X, K, KTT and KTK. Notice that the sum of factors must equal 1.0. If you specify otherwise, each factor will be automatically scaled relatively so that the sum becomes 1.0.



3.3.3.2 Gap classification

You may also specify which gaps to use. The default option is to use those gaps as computed during "Tubular Joint Design". Alternatively you may exclude any gap values present in the model or you can specify a global common gap value by using the option "Manual".



3.3.3.3 Load transfer classification

The "Load Transfer" indicates load transfer through the chord typically used when launch runners. If activated, the effective length of the chord is modified automatically according to the code checking formulas (for API WSD 2002 and API LRFD 2003 only).

3.3.3.4 Through brace classification

The "Through Brace" option. GeniE proposes the through brace in an overlapping joint based on:

1. Maximum thickness is through-brace.
2. Maximum diameter is through, when thicknesses are the same
3. Minimum angle with chord is through brace

3.3.3.5 Weld thickness classification

The "Weld Thickness" option (for API WSD 2002 and API LRFD 2003 only). Used when specifying the weld thickness to the overlapping brace. If "None" is specified the smallest thickness of through or overlapping brace is used.

3.3.3.6 Brace utilisation

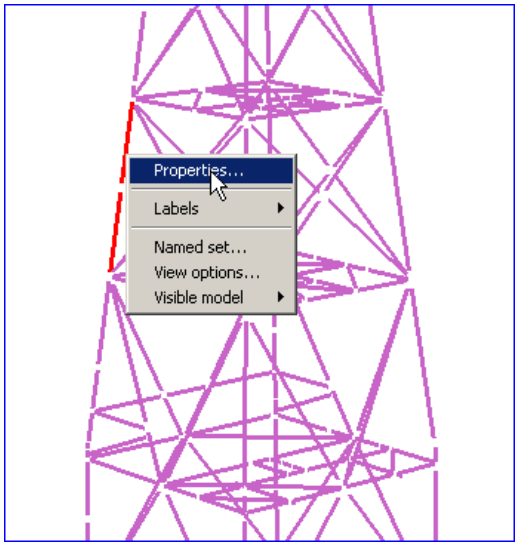
The "Brace utilisation" option is used for ISO 19902 only. This value is used for braces connected to joints classified as critical. For each brace you should select if the brace utilization Ub shall be automatically read from the member check performed, or alternatively give a manually defined brace member usage factor. The usage factor with respect to joint capacity is then checked against (Ub/γ_{zj}). However, note the joint utilization Uj is always scaled with respect to unity, hence for braces connected to a critical joint the utilization factor Uj is multiplied by (γ_{zj}/Ub).
For more information, see the reference document.

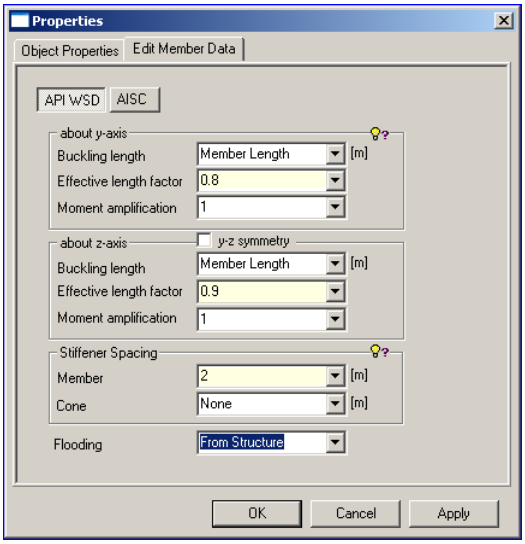
3.4 Local code checking parameters

This Section describes how to modify the parameters for individually capacity members and joints. Common for all is to select a member(s) or joint(s), right mouse button (RMB) and select *Properties*. You can then modify the code checking parameters as described in the previous Chapter.

3.4.1 Modify local member parameters

In the example below, the code checking parameters for the two selected members have been changed compared to the global code checking parameters. The changes are for effective length factors about y-axis and z-axis as well as distance between stiffener spacing.

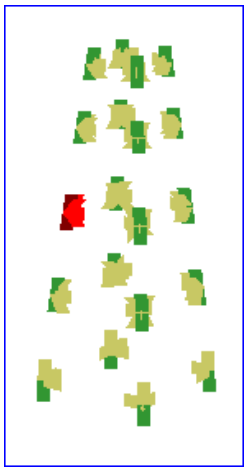


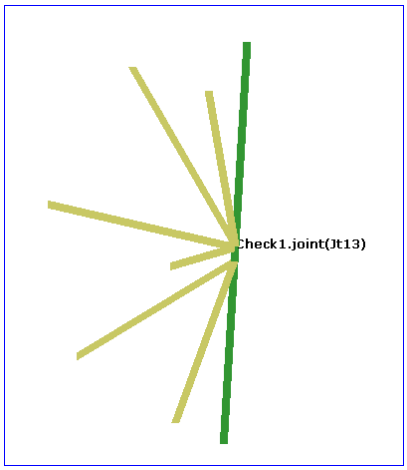


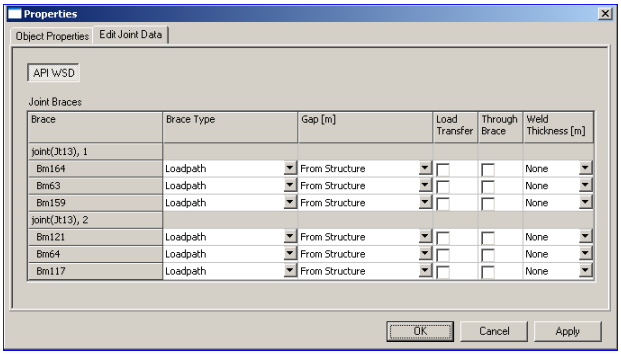
The above examples shows API WSD – the principles for modifying local code checking parameters are the same when it comes to the other code checks supported by GeniE.

3.4.2 Modify local joint parameters

Similarly to modifying local code checking parameters for capacity members you select a joint(s) before modifying any parameter. In the example below one joint has been selected and visualised. The chord is shown with green colour, while the braces are shown with yellow colour.

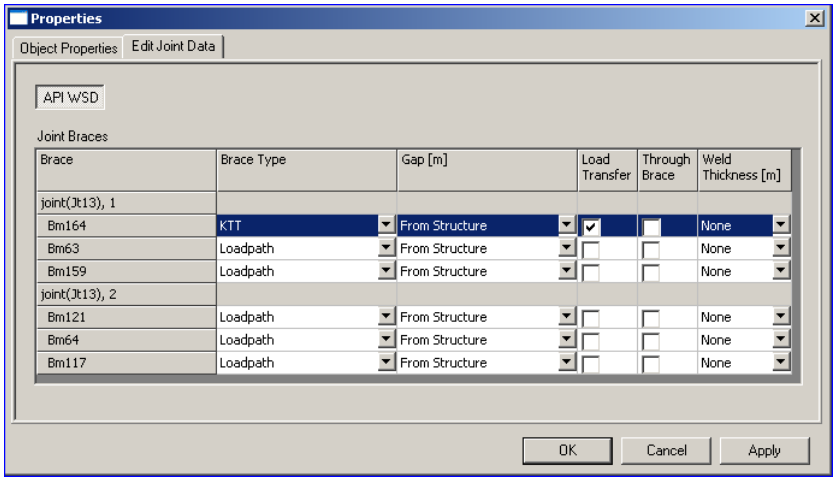


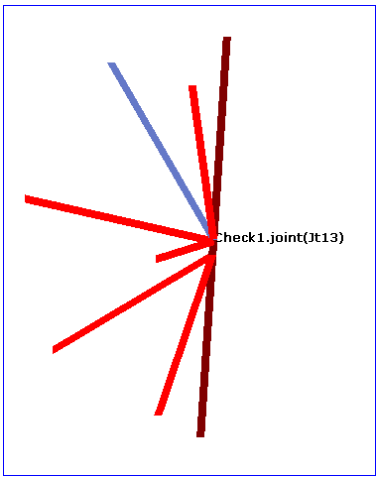


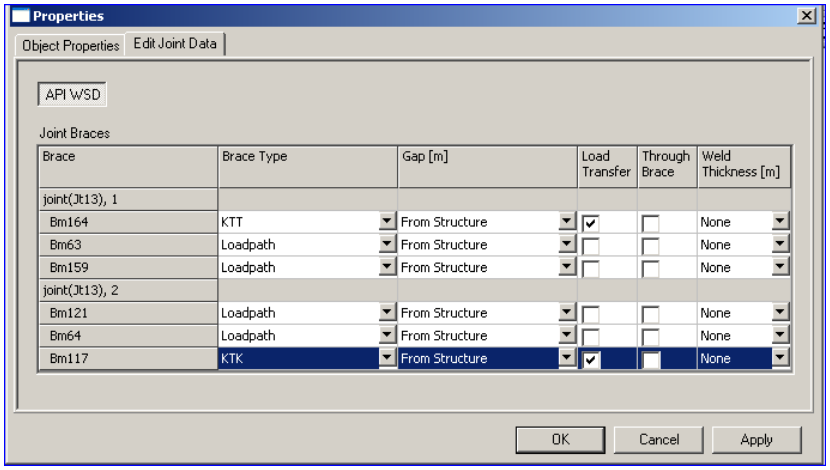


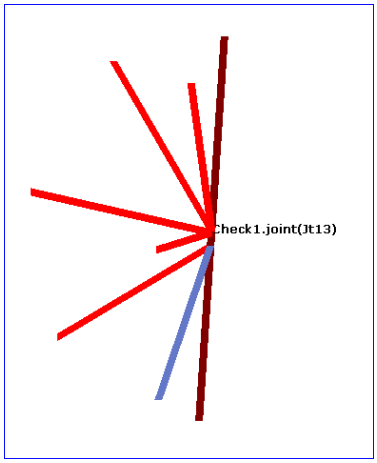
Notice that “Load Transfer” and “Weld Thickness” apply to API WSD only.

To modify the individual braces you need to select them from the brace definition for the actual joint. In the example below two braces have been modified (changes in Brace Type and Load Transfer). Observe that you can graphically identify which brace you are modifying (blue colour).

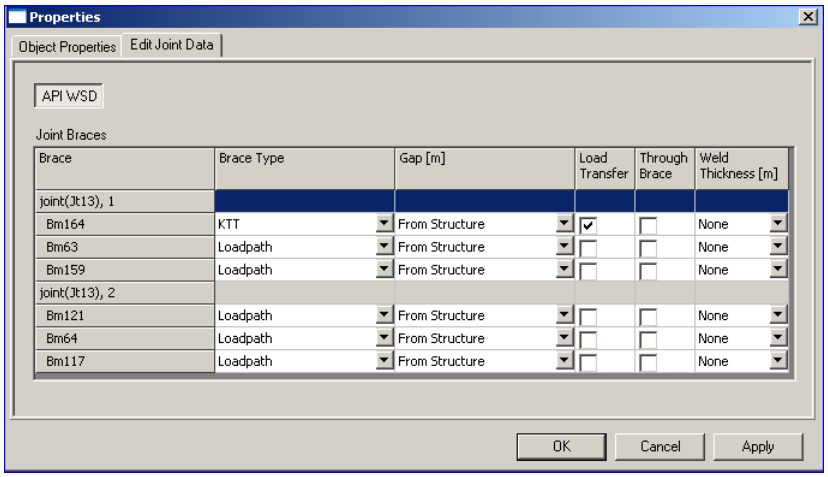


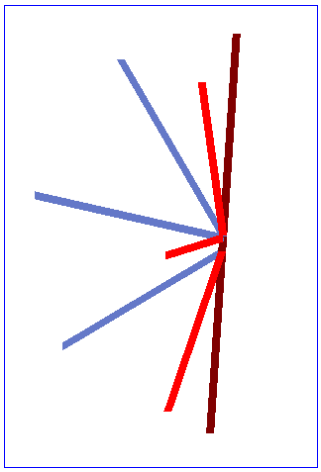






If you click on the joint name in the dialog you will se which braces which are treated to be in the same plane, see below. They are shown with blue colour.





3.5 Code checking positions and forces

The code checking is based on finite element results. For a straight beam, the displacements and forces are computed at the finite element nodes. In addition to first order beams, it is possible to do code checking of straight higher order beams (i.e. beams with 3 finite element nodes).

To do the code checking it is necessary to compute forces at additional positions along the beam:

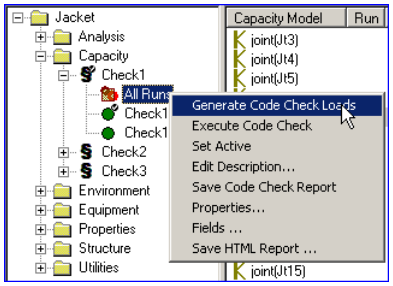
- For regular straight beams: At beam ends, middle position, 25% and 75% positions
- For segmented beams: In addition to above at positions where change in section or material properties
- Moment based positions: Where maximum in-plane or out-of-plane moments occur.

These are described in the following.

The code checking forces and positions are computed by using the command "Generate Code Checking Loads".

Notice when activating the command from the "All Runs" folder, the code checking forces and positions are computed for all runs defined, in this case "Check1.run(1)" and "Check1.run(2)".

Similarly, if you activate the command from the "Check.run(1)" folder the code checking forces and positions are computed for this run only.

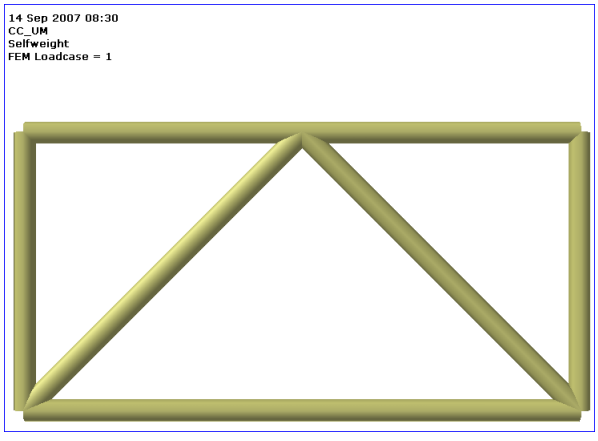


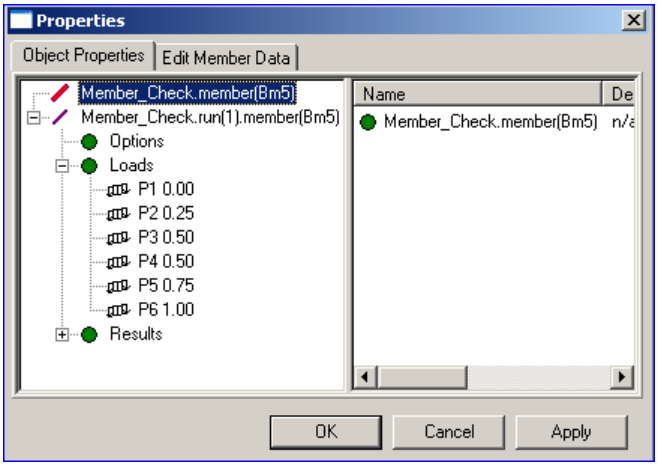
To exemplify the code checking positions, the following models will be used for reference. For all models the capacity models have been created by the option "Split at incoming beams".

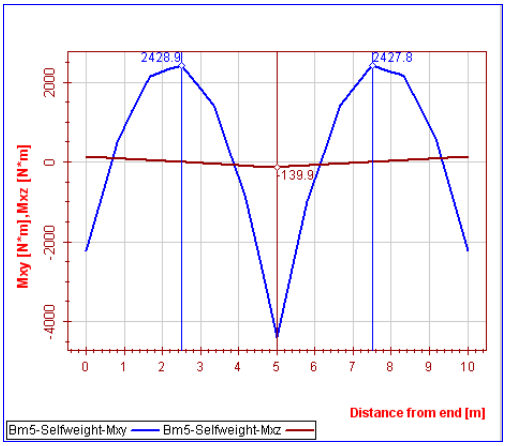
Regular beams having no segments or point/line loads. The beams are loaded with self weight.

The moment distributions (Mxy (in-plane) and Mxz (out-of-plane)) show that peak values occur at pre-defined positions for the upper beam, i.e. at positions 25% and 50%.

The code checking positions for the upper beam (Bm5) are thus at 0%, 25%, 50%, 75% and 100% of the beam length.

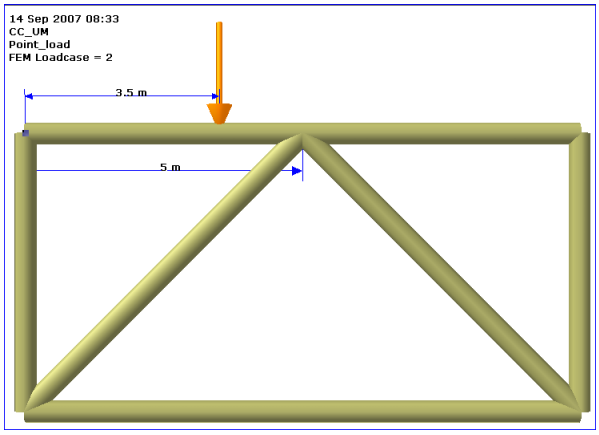




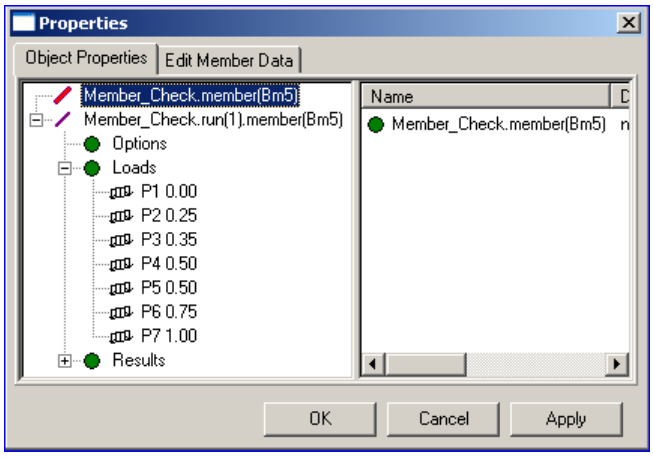


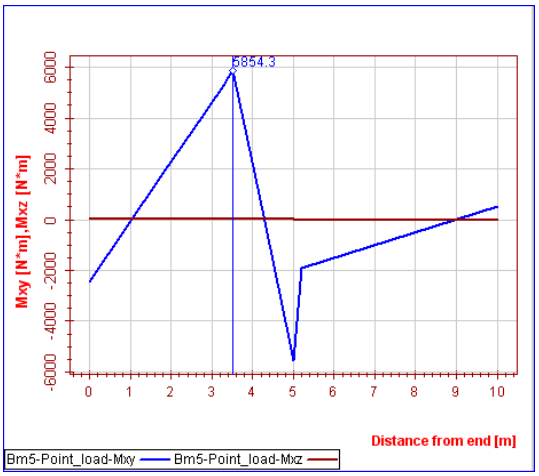
Regular beams having no segments but a point load

The moment distributions (Mxy (in-plane) and Mxz (out-of-plane)) show that maximum values occur at position 3.5 m from the end of beam Bm5 (i.e. at 35%).



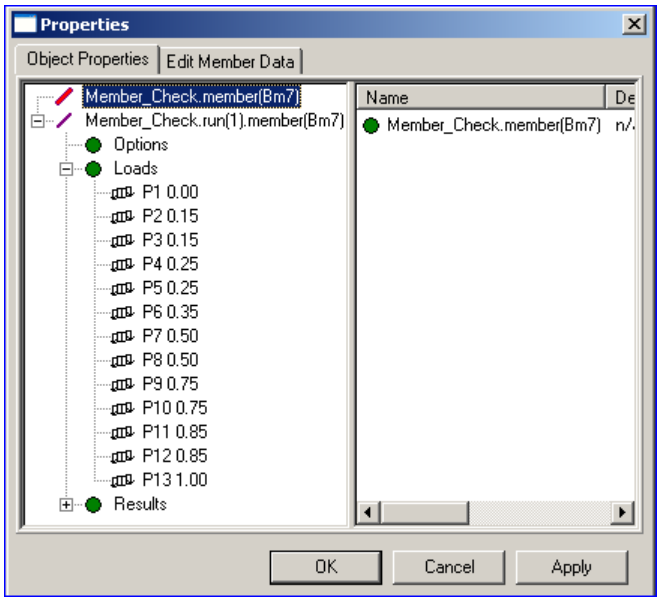
The code checking positions for the upper beam (Bm5) are thus at 0%, 25%, 35%, 50%, 75% and 100% of the beam length:

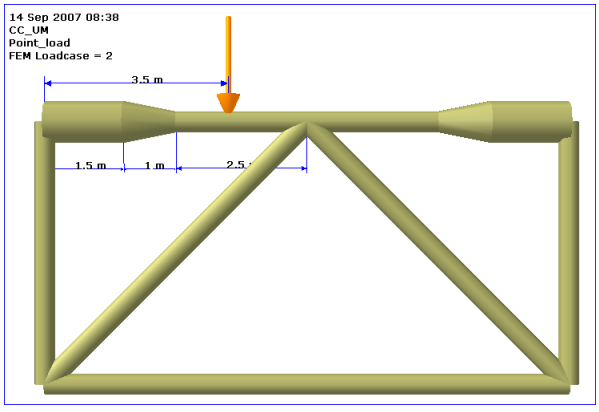




The segmented beam has a point load.

In addition to the maximum occurring moment the segmented positions are taken into consideration.



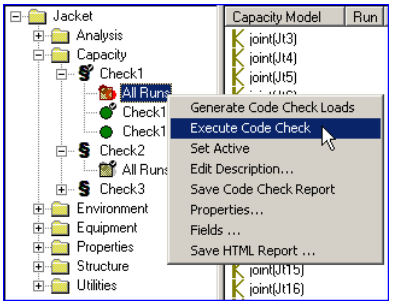


As can be seen additional code checking positions are added on both sides of the segmented positions, typically at both sides at 15% (P2 and P3).

Please notice you learn how to document code checking positions and forces in Chapter 3.7 “Investigate the results”.

3.6 Perform the code check

When the code check positions and forces have been computed it is possible to do the code checking. Multiple code checks can be performed for the various runs at the same time or you may decide to do it for individual code check runs.



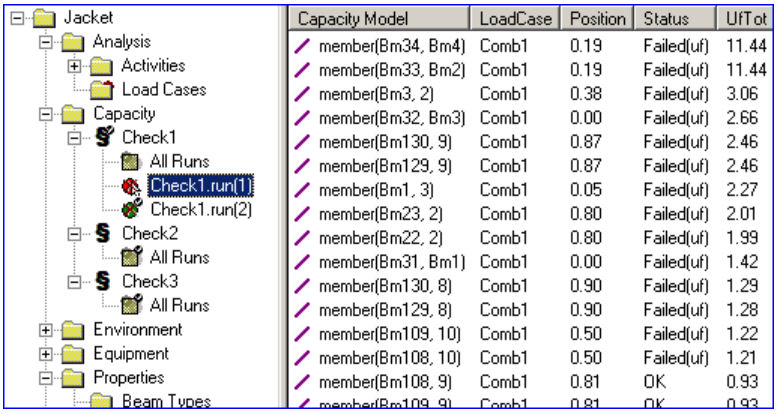
The code check is executed by using the command "Execute Code Check".

Notice when activating the command from the "All Runs" folder, the code check is performed for all runs defined, in this case "Check1.run(1)" and "Check1.run(2)".

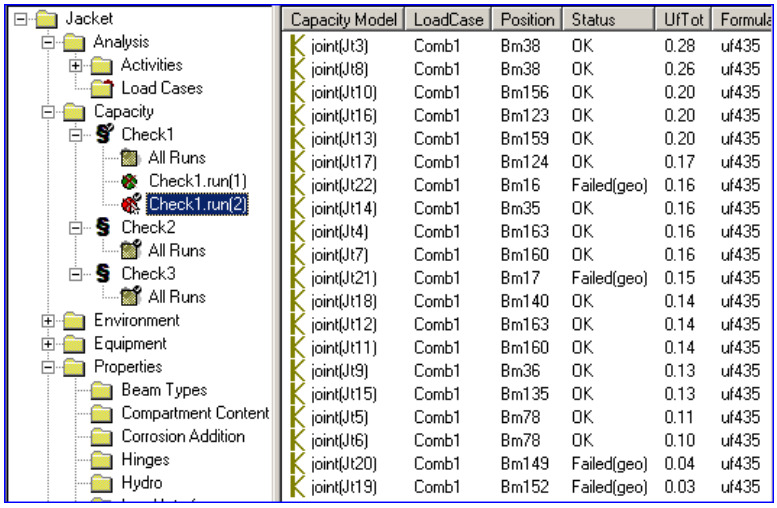
Similarly, if you activate the command from the "Check.run(1)" folder the code check is computed for this run only.

When the code check has been performed the browser includes results from the code check. The examples below show a typical browser view for a member check and a joint check (remember to specify which run is active if you have separate runs for member and joint check). Notice that the code checks examples have been executed using program default values.

An example of viewing member code check results in the browser is seen to the right.


Jacket
Analysis
Activities
Load Cases
Capacity
Check1
All Runs
Check1.run(1)
Check1.run(2)
Check2
All Runs
Check3
All Runs
Environment
Equipment
Properties
Beam Types
Capacity Model | LoadCase | Position | Status | UfTot
member(Bm34, Bm4) | Comb1 | 0.19 | Failed(uf) | 11.44
member(Bm33, Bm2) | Comb1 | 0.19 | Failed(uf) | 11.44
member(Bm3, 2) | Comb1 | 0.38 | Failed(uf) | 3.06
member(Bm32, Bm3) | Comb1 | 0.00 | Failed(uf) | 2.66
member(Bm130, 9) | Comb1 | 0.87 | Failed(uf) | 2.46
member(Bm129, 9) | Comb1 | 0.87 | Failed(uf) | 2.46
member(Bm1, 3) | Comb1 | 0.05 | Failed(uf) | 2.27
member(Bm23, 2) | Comb1 | 0.80 | Failed(uf) | 2.01
member(Bm22, 2) | Comb1 | 0.80 | Failed(uf) | 1.99
member(Bm31, Bm1) | Comb1 | 0.00 | Failed(uf) | 1.42
member(Bm130, 8) | Comb1 | 0.90 | Failed(uf) | 1.29
member(Bm129, 8) | Comb1 | 0.90 | Failed(uf) | 1.28
member(Bm109, 10) | Comb1 | 0.50 | Failed(uf) | 1.22
member(Bm108, 10) | Comb1 | 0.50 | Failed(uf) | 1.21
member(Bm108, 9) | Comb1 | 0.81 | OK | 0.93

Similarly, results from a joint check are shown to the right.


Jacket
Analysis
Activities
Load Cases
Capacity
Check1
All Runs
Check1.run(1)
Check1.run(2)
Check2
All Runs
Check3
All Runs
Environment
Equipment
Properties
Beam Types
Compartment Content
Corrosion Addition
Hinges
Hydro
Capacity Model | LoadCase | Position | Status | UfTot | Formula
joint(Jt3) | Comb1 | Bm38 | OK | 0.28 | uf435
joint(Jt8) | Comb1 | Bm38 | OK | 0.26 | uf435
joint(Jt10) | Comb1 | Bm156 | OK | 0.20 | uf435
joint(Jt16) | Comb1 | Bm123 | OK | 0.20 | uf435
joint(Jt13) | Comb1 | Bm159 | OK | 0.20 | uf435
joint(Jt17) | Comb1 | Bm124 | OK | 0.17 | uf435
joint(Jt22) | Comb1 | Bm16 | Failed(geo) | 0.16 | uf435
joint(Jt14) | Comb1 | Bm35 | OK | 0.16 | uf435
joint(Jt4) | Comb1 | Bm163 | OK | 0.16 | uf435
joint(Jt7) | Comb1 | Bm160 | OK | 0.16 | uf435
joint(Jt21) | Comb1 | Bm17 | Failed(geo) | 0.15 | uf435
joint(Jt18) | Comb1 | Bm140 | OK | 0.14 | uf435
joint(Jt12) | Comb1 | Bm163 | OK | 0.14 | uf435
joint(Jt11) | Comb1 | Bm160 | OK | 0.14 | uf435
joint(Jt9) | Comb1 | Bm36 | OK | 0.13 | uf435
joint(Jt15) | Comb1 | Bm135 | OK | 0.13 | uf435
joint(Jt5) | Comb1 | Bm78 | OK | 0.11 | uf435
joint(Jt6) | Comb1 | Bm78 | OK | 0.10 | uf435
joint(Jt20) | Comb1 | Bm149 | Failed(geo) | 0.04 | uf435
joint(Jt19) | Comb1 | Bm152 | Failed(geo) | 0.03 | uf435

The next Chapter explains how to find and present the various code checking results.

3.7 Investigate the results

There are four ways of investigating code checking results – see

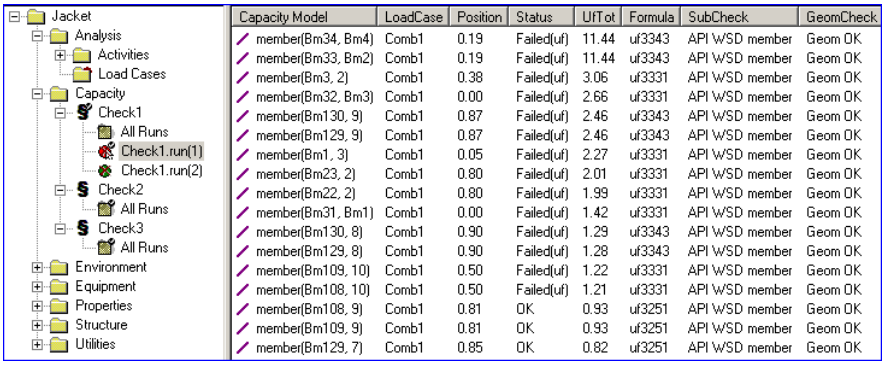
- From the browser
- Graphically
- From object property
- A report

The three first alternatives are described in this Chapter while Chapter 3.9 “make a report” shows how to make a default report or how to customize a report.

3.7.1 From the browser

3.7.1.1 Member code check

The default view is set up so that it sorts on the highest utilisation factor (UfTot). The results shown are dependent on which load case you have set to active.

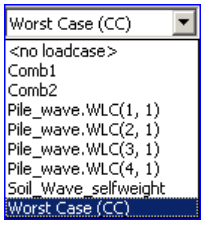


Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
member(Bm34, Bm4)	Comb1	0.19	Failed(uf)	11.44	uf3343	API WSD member	Geom OK
member(Bm33, Bm2)	Comb1	0.19	Failed(uf)	11.44	uf3343	API WSD member	Geom OK
member(Bm3, 2)	Comb1	0.38	Failed(uf)	3.06	uf3331	API WSD member	Geom OK
member(Bm32, Bm3)	Comb1	0.00	Failed(uf)	2.66	uf3331	API WSD member	Geom OK
member(Bm130, 9)	Comb1	0.87	Failed(uf)	2.46	uf3343	API WSD member	Geom OK
member(Bm129, 9)	Comb1	0.87	Failed(uf)	2.46	uf3343	API WSD member	Geom OK
member(Bm1, 3)	Comb1	0.05	Failed(uf)	2.27	uf3331	API WSD member	Geom OK
member(Bm23, 2)	Comb1	0.80	Failed(uf)	2.01	uf3331	API WSD member	Geom OK
member(Bm22, 2)	Comb1	0.80	Failed(uf)	1.99	uf3331	API WSD member	Geom OK
member(Bm31, Bm1)	Comb1	0.00	Failed(uf)	1.42	uf3331	API WSD member	Geom OK
member(Bm130, 8)	Comb1	0.90	Failed(uf)	1.29	uf3343	API WSD member	Geom OK
member(Bm129, 8)	Comb1	0.90	Failed(uf)	1.28	uf3343	API WSD member	Geom OK
member(Bm109, 10)	Comb1	0.50	Failed(uf)	1.22	uf3331	API WSD member	Geom OK
member(Bm108, 10)	Comb1	0.50	Failed(uf)	1.21	uf3331	API WSD member	Geom OK
member(Bm108, 9)	Comb1	0.81	OK	0.93	uf3251	API WSD member	Geom OK
member(Bm109, 9)	Comb1	0.81	OK	0.93	uf3251	API WSD member	Geom OK
member(Bm129, 7)	Comb1	0.85	OK	0.82	uf3251	API WSD member	Geom OK

As can be seen, there are several failures because of the code check while all the geometry checks satisfy the necessary criteria.

In addition the relative position for the worst code check result for each capacity member is shown.

If you specify the option Worst Case (CC) from the load pulldown list, the browser will find the worst condition for each code checking position and report it.



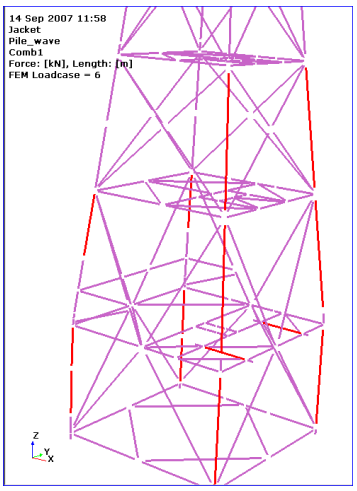
The “Formula” column identifies which formula was used to do the code checking. This is a reference to the relevant formula as listed in the various codes of practice supported by GeniE.

The column for “SubCheck” lists which type of code check that has been performed, i.e. a pipe section check or a non-pipe section check.

You may sort each column in the browser by clicking on the column header.

If you select some of the capacity members with a high utilisation factor it is easy to graphically to see where they occur.

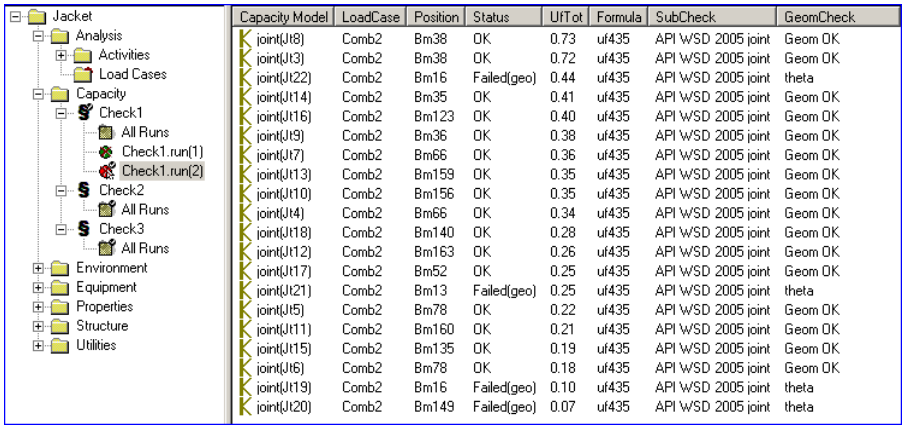
Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
member(Bm34, Bm4)	Comb1	0.19	Failed(uf)	11.44	uf3343	API WSD member	Geom OK
member(Bm33, Bm2)	Comb1	0.19	Failed(uf)	11.44	uf3343	API WSD member	Geom OK
member(Bm3, 2)	Comb1	0.38	Failed(uf)	3.06	uf3331	API WSD member	Geom OK
member(Bm32, Bm3)	Comb1	0.00	Failed(uf)	2.66	uf3331	API WSD member	Geom OK
member(Bm130, 9)	Comb1	0.87	Failed(uf)	2.46	uf3343	API WSD member	Geom OK
member(Bm129, 9)	Comb1	0.87	Failed(uf)	2.46	uf3343	API WSD member	Geom OK
member(Bm1, 3)	Comb1	0.05	Failed(uf)	2.27	uf3331	API WSD member	Geom OK
member(Bm23, 2)	Comb1	0.80	Failed(uf)	2.01	uf3331	API WSD member	Geom OK
member(Bm22, 2)	Comb1	0.80	Failed(uf)	1.99	uf3331	API WSD member	Geom OK
member(Bm31, Bm1)	Comb1	0.00	Failed(uf)	1.42	uf3331	API WSD member	Geom OK
member(Bm130, 8)	Comb1	0.90	Failed(uf)	1.29	uf3343	API WSD member	Geom OK
member(Bm129, 8)	Comb1	0.90	Failed(uf)	1.28	uf3343	API WSD member	Geom OK
member(Bm109, 10)	Comb1	0.50	Failed(uf)	1.22	uf3331	API WSD member	Geom OK
member(Bm108, 10)	Comb1	0.50	Failed(uf)	1.21	uf3331	API WSD member	Geom OK
member(Bm108, 9)	Comb1	0.81	OK	0.93	uf3251	API WSD member	Geom OK
member(Bm109, 9)	Comb1	0.81	OK	0.93	uf3251	API WSD member	Geom OK
member(Bm129, 7)	Comb1	0.85	OK	0.82	uf3251	API WSD member	Geom OK
member(Bm130, 7)	Comb1	0.85	OK	0.82	uf3251	API WSD member	Geom OK



3.7.1.2 Joint check

The default view is set up so that it sorts on the highest utilisation factor (UfTot). The results shown are dependent on which load case you have set to active.

As can be seen, there are several failures because of the geometry check while all the code checks satisfy the necessary criteria.

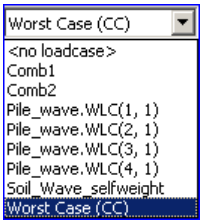


Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
joint(Jt8)	Comb2	Bm38	OK	0.73	uf435	API WSD 2005 joint	Geom OK
joint(Jt3)	Comb2	Bm38	OK	0.72	uf435	API WSD 2005 joint	Geom OK
joint(Jt22)	Comb2	Bm16	Failed(geo)	0.44	uf435	API WSD 2005 joint	theta
joint(Jt14)	Comb2	Bm35	OK	0.41	uf435	API WSD 2005 joint	Geom OK
joint(Jt16)	Comb2	Bm123	OK	0.40	uf435	API WSD 2005 joint	Geom OK
joint(Jt9)	Comb2	Bm36	OK	0.38	uf435	API WSD 2005 joint	Geom OK
joint(Jt7)	Comb2	Bm66	OK	0.36	uf435	API WSD 2005 joint	Geom OK
joint(Jt13)	Comb2	Bm159	OK	0.35	uf435	API WSD 2005 joint	Geom OK
joint(Jt10)	Comb2	Bm156	OK	0.35	uf435	API WSD 2005 joint	Geom OK
joint(Jt4)	Comb2	Bm66	OK	0.34	uf435	API WSD 2005 joint	Geom OK
joint(Jt18)	Comb2	Bm140	OK	0.28	uf435	API WSD 2005 joint	Geom OK
joint(Jt12)	Comb2	Bm163	OK	0.26	uf435	API WSD 2005 joint	Geom OK
joint(Jt17)	Comb2	Bm52	OK	0.25	uf435	API WSD 2005 joint	Geom OK
joint(Jt21)	Comb2	Bm13	Failed(geo)	0.25	uf435	API WSD 2005 joint	theta
joint(Jt5)	Comb2	Bm78	OK	0.22	uf435	API WSD 2005 joint	Geom OK
joint(Jt11)	Comb2	Bm160	OK	0.21	uf435	API WSD 2005 joint	Geom OK
joint(Jt15)	Comb2	Bm135	OK	0.19	uf435	API WSD 2005 joint	Geom OK
joint(Jt6)	Comb2	Bm78	OK	0.18	uf435	API WSD 2005 joint	Geom OK
joint(Jt19)	Comb2	Bm16	Failed(geo)	0.10	uf435	API WSD 2005 joint	theta
joint(Jt20)	Comb2	Bm149	Failed(geo)	0.07	uf435	API WSD 2005 joint	theta

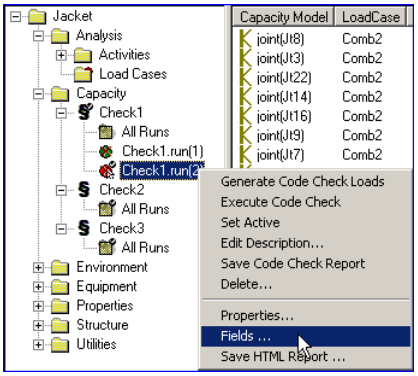
In addition the brace that has the worst code check result for each joint is shown.

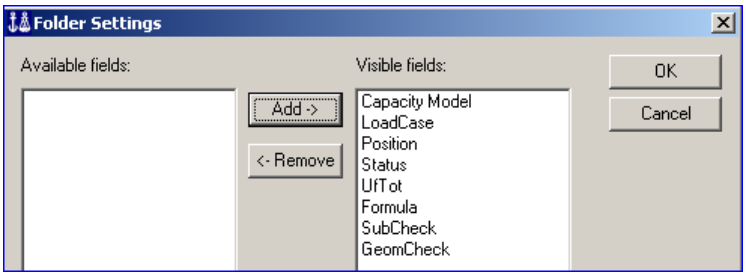
If you specify the option Worst Case (CC) from the load pulldown list, the browser will find the worst condition for each code checking position and report it.

The rest of the options are similar to those for the member check.



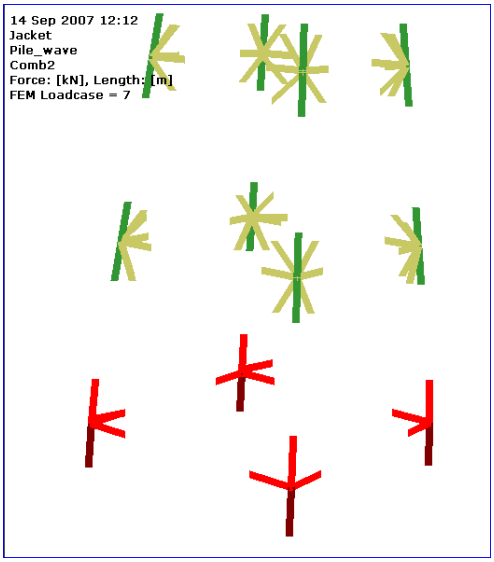
You may also customise the browser view by selecting “Fields” from a code check run folder.





The example below shows how to graphically visualise those joints failing in the geometry check (in this case the browser is set to sort on geometry check).

Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
joint(Jt22)	Comb2	Bm16	Failed(geo)	0.44	uf435	API WSD 2005 joint	theta
joint(Jt21)	Comb2	Bm13	Failed(geo)	0.25	uf435	API WSD 2005 joint	theta
joint(Jt19)	Comb2	Bm16	Failed(geo)	0.10	uf435	API WSD 2005 joint	theta
joint(Jt20)	Comb2	Bm149	Failed(geo)	0.07	uf435	API WSD 2005 joint	theta
joint(Jt8)	Comb2	Bm38	OK	0.73	uf435	API WSD 2005 joint	Geom OK
joint(Jt3)	Comb2	Bm38	OK	0.72	uf435	API WSD 2005 joint	Geom OK
joint(Jt14)	Comb2	Bm35	OK	0.41	uf435	API WSD 2005 joint	Geom OK
joint(Jt16)	Comb2	Bm123	OK	0.40	uf435	API WSD 2005 joint	Geom OK
joint(Jt9)	Comb2	Bm36	OK	0.38	uf435	API WSD 2005 joint	Geom OK
joint(Jt7)	Comb2	Bm66	OK	0.36	uf435	API WSD 2005 joint	Geom OK
joint(Jt13)	Comb2	Bm159	OK	0.35	uf435	API WSD 2005 joint	Geom OK
joint(Jt10)	Comb2	Bm156	OK	0.35	uf435	API WSD 2005 joint	Geom OK
joint(Jt4)	Comb2	Bm66	OK	0.34	uf435	API WSD 2005 joint	Geom OK
joint(Jt18)	Comb2	Bm140	OK	0.28	uf435	API WSD 2005 joint	Geom OK
joint(Jt12)	Comb2	Bm163	OK	0.26	uf435	API WSD 2005 joint	Geom OK
joint(Jt17)	Comb2	Bm52	OK	0.25	uf435	API WSD 2005 joint	Geom OK
joint(Jt5)	Comb2	Bm78	OK	0.22	uf435	API WSD 2005 joint	Geom OK
joint(Jt11)	Comb2	Bm160	OK	0.21	uf435	API WSD 2005 joint	Geom OK
joint(Jt15)	Comb2	Bm135	OK	0.19	uf435	API WSD 2005 joint	Geom OK
joint(Jt6)	Comb2	Bm78	OK	0.18	uf435	API WSD 2005 joint	Geom OK



3.7.2 From the graphic window.

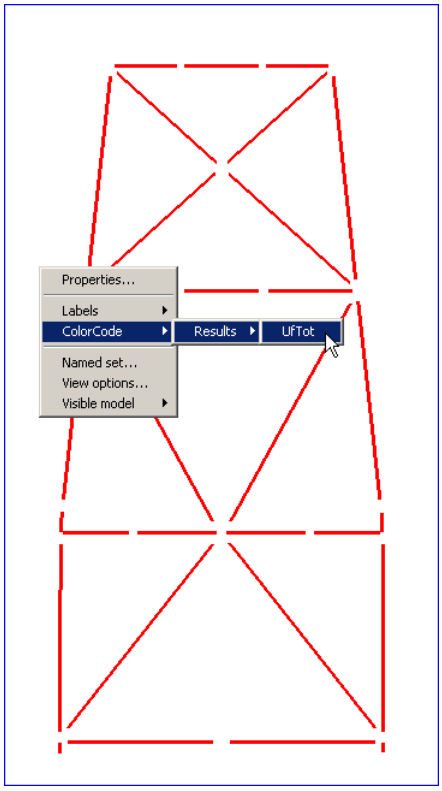
You may select and visualise parts of the capacity model similar to how you do this for other objects like beams, plates, loads etc. In the following parts of the capacity model has been selected.

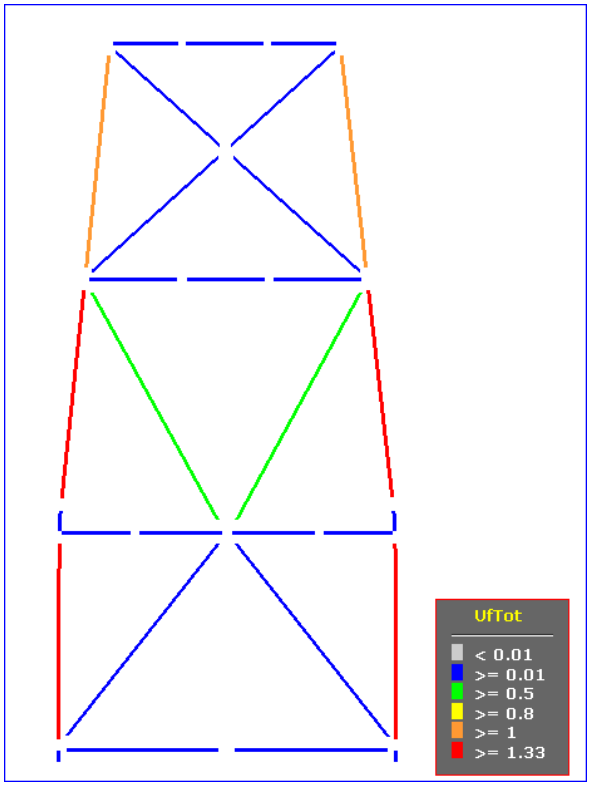
3.7.2.1 Member code check

Graphic presentation of results depends on the load case that is selected. You may also select the Worst Case (CC) to display the worst results for all load cases part of the code check.

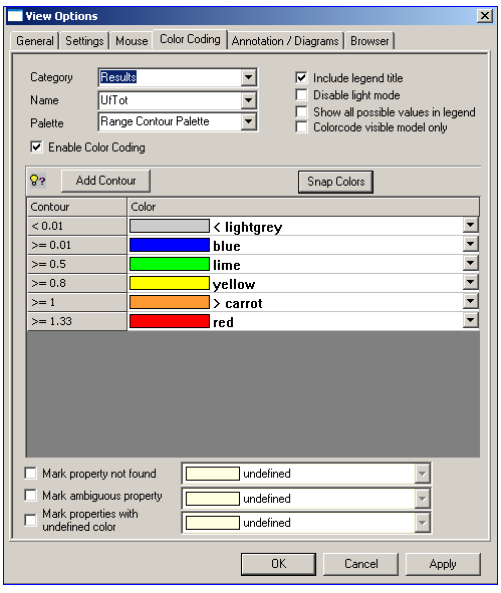
In the following there are examples on how to present results. Not all are shown herein, but all relevant commands are shown.

Colour coding of utilisation factors from code check.





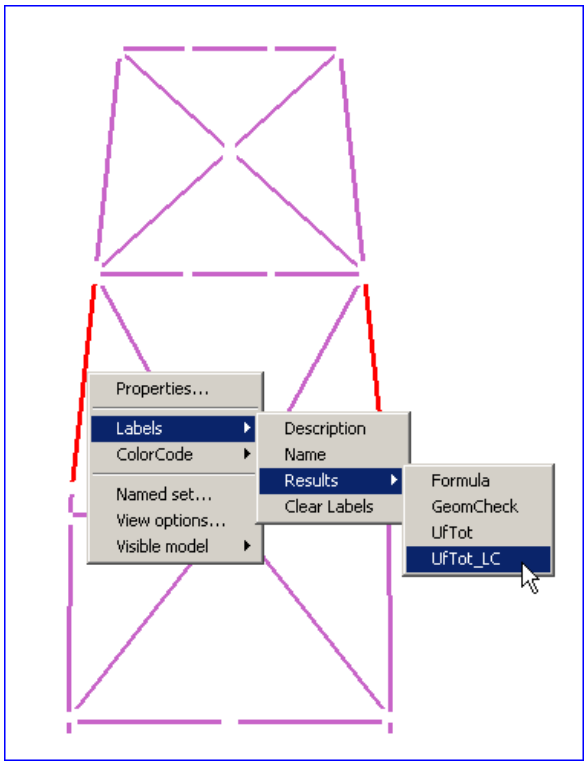
The colour coding is based on default thresholds. You may customise these settings (both the colours, the threshold values as well as number of thresholds) from the command View|Options|Color Coding.

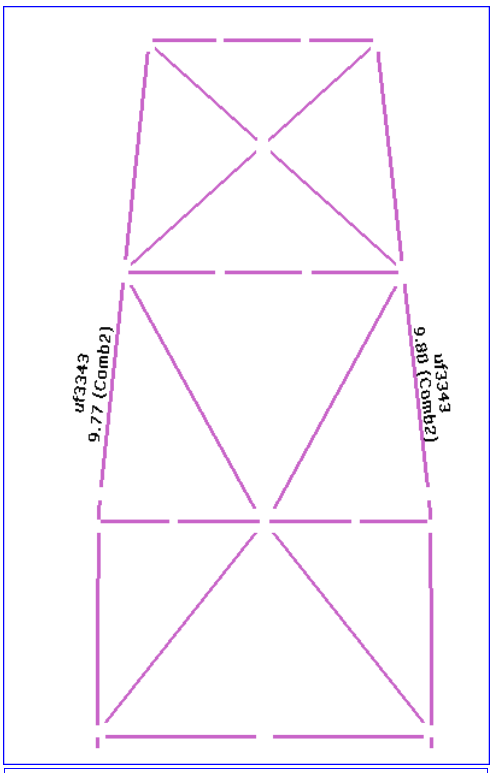


Labelling results from code check.

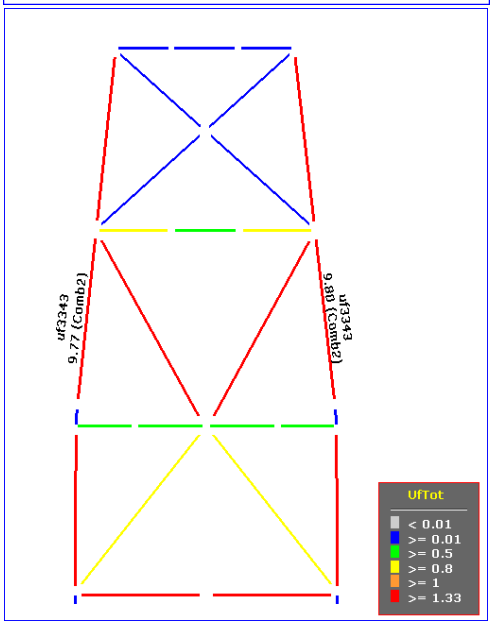
To add a label to an object (a capacity member in this case) you need to select the object and right click to see the available labelling options. You may add labels as shown in the picture below. It is also possible to have several labels on the same object; the example below shows the utilisation factor as well as the formula used during the code checking.

The label UfTot_LC shows the utilisation factor and relevant load case.





It is also possible to do colour coding of utilisation factors and add labels. In the example to the right the above colour coding has been added.



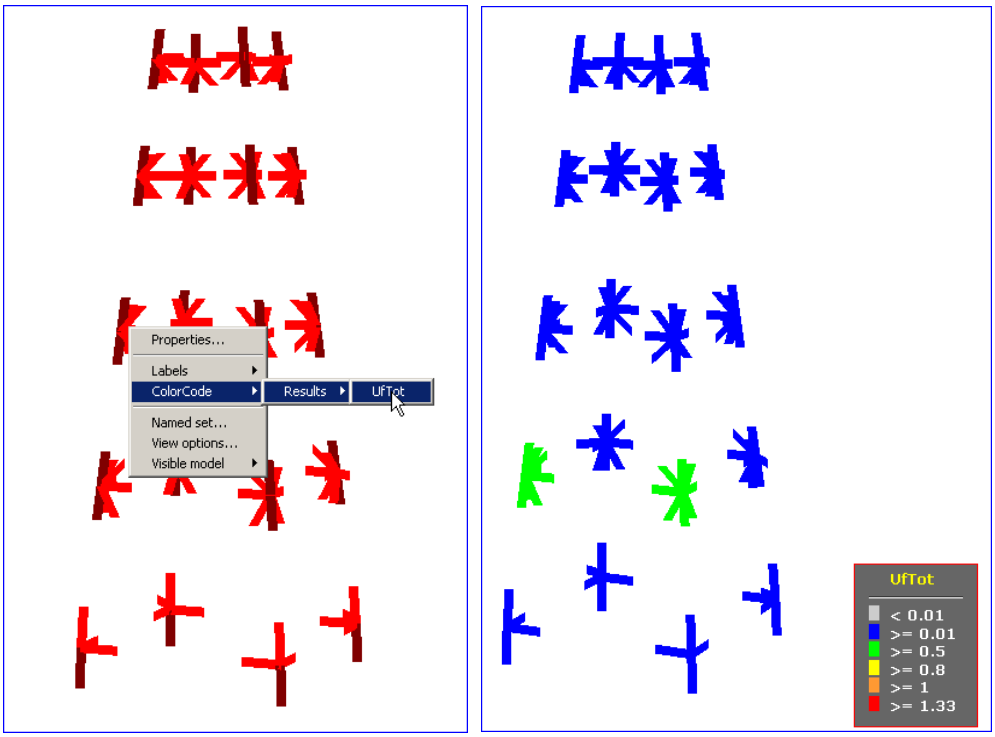
3.7.2.2 Joint check

Graphic presentation of results depends on the load case that is selected. You may also select the Worst Case (CC) to display the worst results for all load cases part of the joint check.

In the following there are examples on how to present results. Not all are shown herein, but all relevant commands are shown.

Colour coding of utilisation factors from joint check.

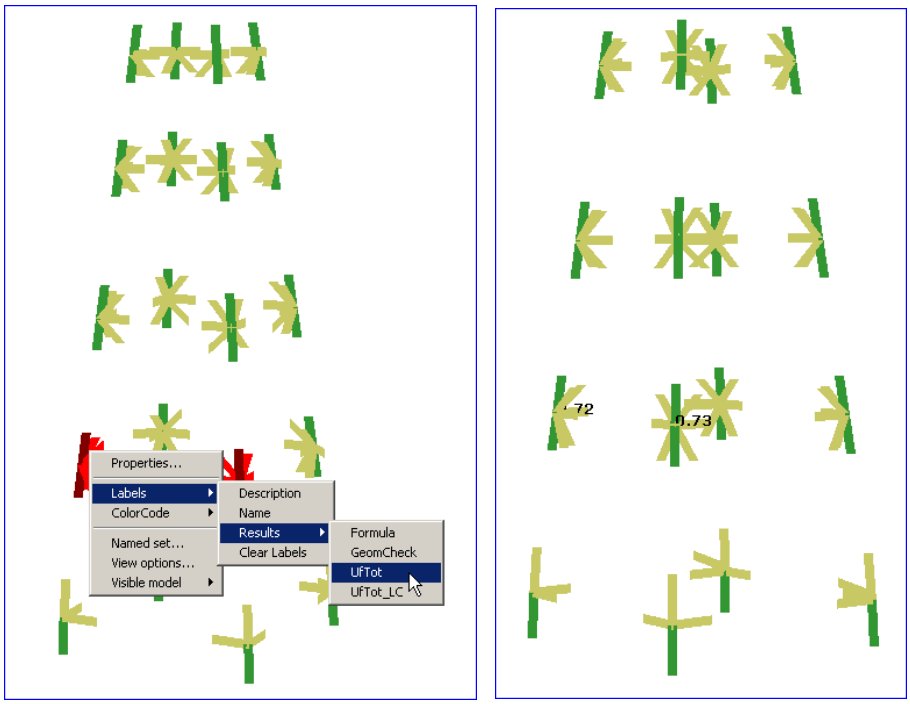
You may also manipulate the thresholds as shown for colour coding of member utilisation factors.



Labelling results from joint check.

The example to the right shows utilisation factors for two joints.

It is also possible to add colour coding of utilisation factors.



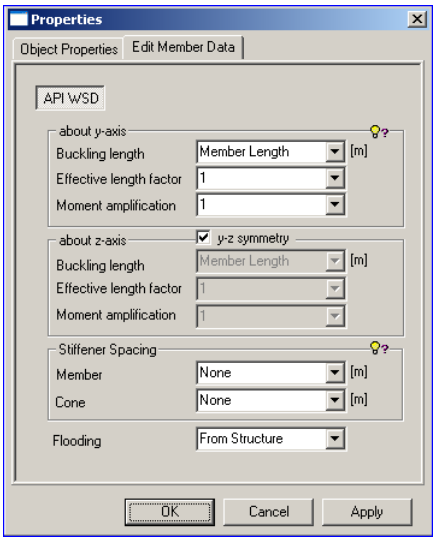
3.7.3 From object property

The previous ways of documenting the results are primarily meant to give you an overview on whether the code checks are within satisfactory levels or not. If a member fails it is necessary to investigate the code checking results more detailed to decide how to strengthen the member. Detailed code checking results may be found from the object browser or from a printed report. Both options are explained in the following.

It may also be that you need to re-distribute the forces in the structure to improve the code checking results. To understand the moment and force distribution you may use the features for visualising these either in a 3 dimensional view (Tools|Analysis|Presentation) or in a 2 dimensional view (Tools|Analysis|Beam Result Diagram).

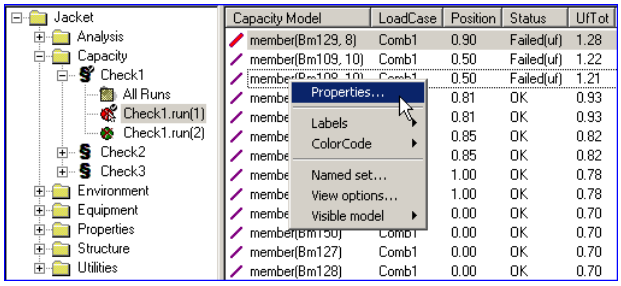
3.7.3.1 Detailed results for member check.

To access detailed results from the object property you select the member – either from browser or graphically - you want to investigate, RMB and select *Properties*.



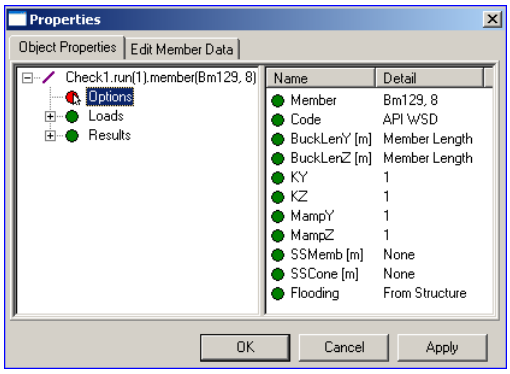
The member data are listed as shown to the right. Please observe that GeniE detects whether the member has a tubular section property (API WSD and NORSOK) or a non-tubular profile (AISC and EUROCODE).

You may also modify member data for later use.



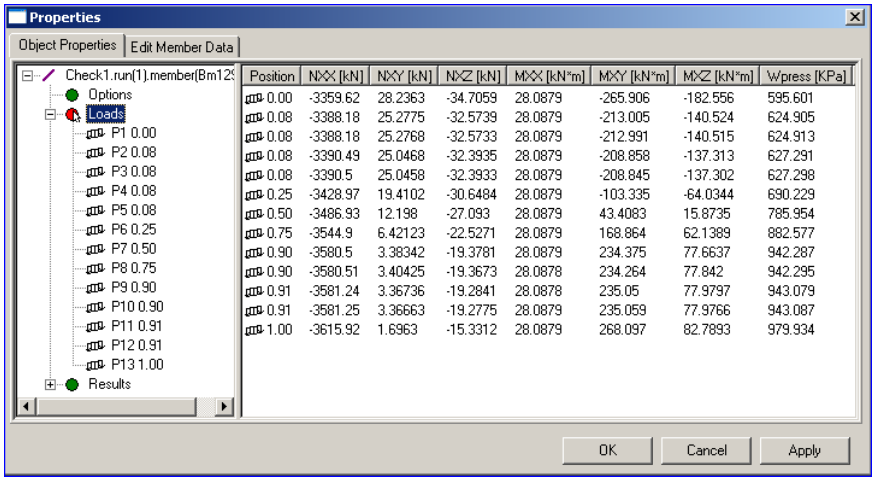
Member properties - options.

The Options folder for the Object Properties lists which options have been assigned to the capacity member. In this case default values have been used. The options to use are described in the previous *Chapter 3.3.2 Define global member parameters* and *Chapter 3.4.2 Modify local member parameters*.

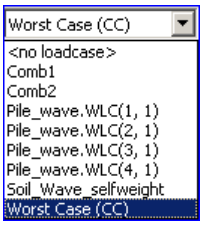


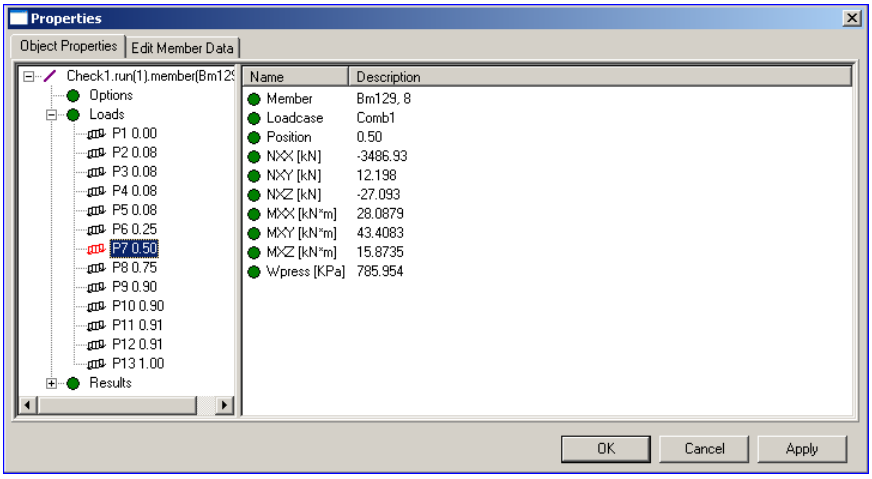
Member properties – code checking forces.

There are two ways of investigating the code checking forces. The alternative to the right gives the worst forces and moments at each code checking position determined by the program. Notice that this member also has water pressure (W_{press}) since it is submerged. The water pressure is automatically accounted for when the environmental conditions are referred to in the analysis runs.



The other option is to investigate the forces and moments per position. In this case the worst load case name is also listed. Notice that the load case shown depends on whether you have set a specific load case to active or used the option *Worst Case (CC)*.

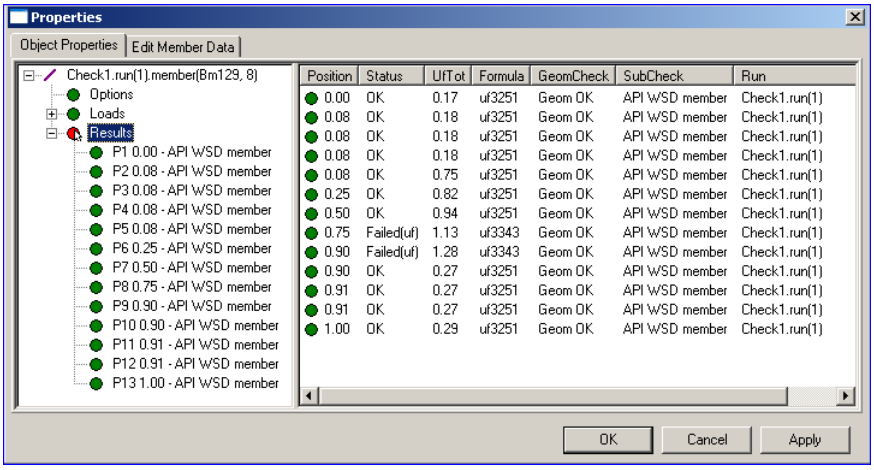




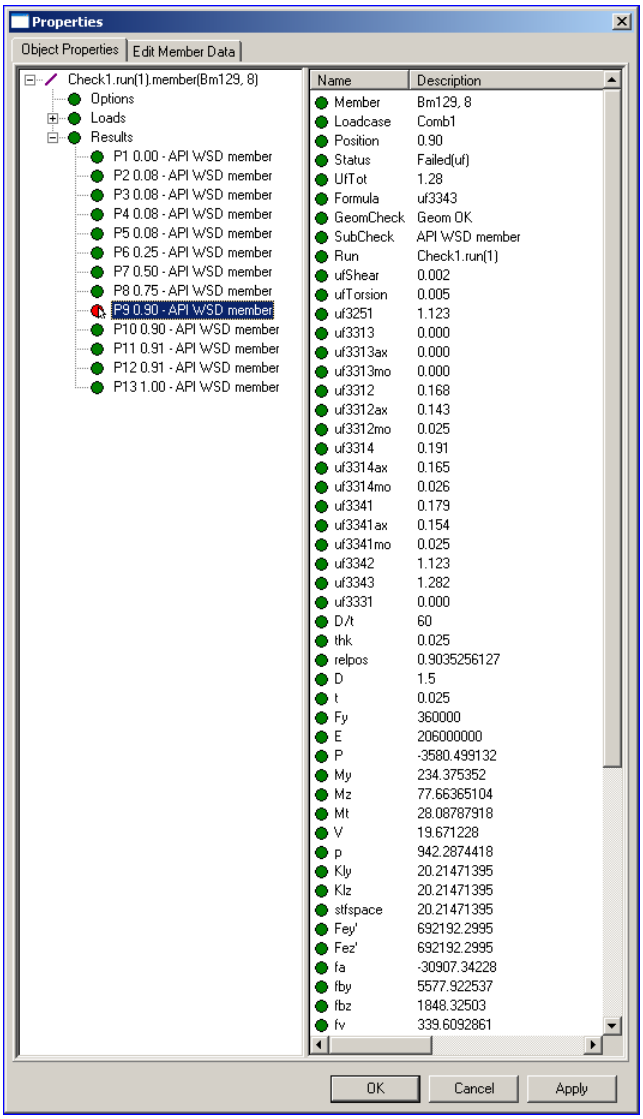
Member properties – code checking results.

The code checking results overview lists the utilisation factors and the status of the geometry check at each code checking position.

In this case all geometry checks are within satisfactory limits, while the code check has failed for two positions (positions 0.75 and 0.90). For both of these positions, the formula uf3343 from the API WSD code of practice was used to compute the utilisation factors.


Properties
Object Properties
Edit Member Data
Check1.run(1).member(Bm129, 8)
Options
Loads
Results
P1 0.00 - API WSD member
P2 0.08 - API WSD member
P3 0.08 - API WSD member
P4 0.08 - API WSD member
P5 0.08 - API WSD member
P6 0.25 - API WSD member
P7 0.50 - API WSD member
P8 0.75 - API WSD member
P9 0.90 - API WSD member
P10 0.90 - API WSD member
P11 0.91 - API WSD member
P12 0.91 - API WSD member
P13 1.00 - API WSD member
Position Status UfTot Formula GeomCheck SubCheck Run
0.00 OK 0.17 uf3251 Geom OK API WSD member Check1.run(1)
0.08 OK 0.18 uf3251 Geom OK API WSD member Check1.run(1)
0.08 OK 0.18 uf3251 Geom OK API WSD member Check1.run(1)
0.08 OK 0.18 uf3251 Geom OK API WSD member Check1.run(1)
0.08 OK 0.75 uf3251 Geom OK API WSD member Check1.run(1)
0.25 OK 0.82 uf3251 Geom OK API WSD member Check1.run(1)
0.50 OK 0.94 uf3251 Geom OK API WSD member Check1.run(1)
0.75 Failed(uf) 1.13 uf3343 Geom OK API WSD member Check1.run(1)
0.90 Failed(uf) 1.28 uf3343 Geom OK API WSD member Check1.run(1)
0.90 OK 0.27 uf3251 Geom OK API WSD member Check1.run(1)
0.91 OK 0.27 uf3251 Geom OK API WSD member Check1.run(1)
0.91 OK 0.27 uf3251 Geom OK API WSD member Check1.run(1)
1.00 OK 0.29 uf3251 Geom OK API WSD member Check1.run(1)
OK
Cancel
Apply

The detailed view of code checking results (in this case at position 0.90) lists all the attributes that have been computed during code check. For a full documentation of the nomenclature, please see *Appendix B Code checks and nomenclature*.

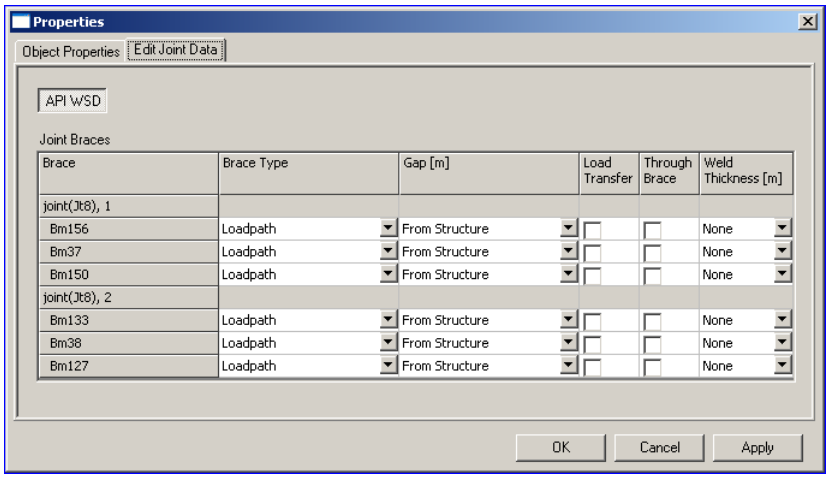


3.7.3.2 Detailed results for joint check.

To access detailed results from the object property you select the joint – either from browser or graphically - you want to investigate, RMB and select *Properties*.

The member data are listed as shown to the right. In this case the joint consists of two joint checks representing two different planes.

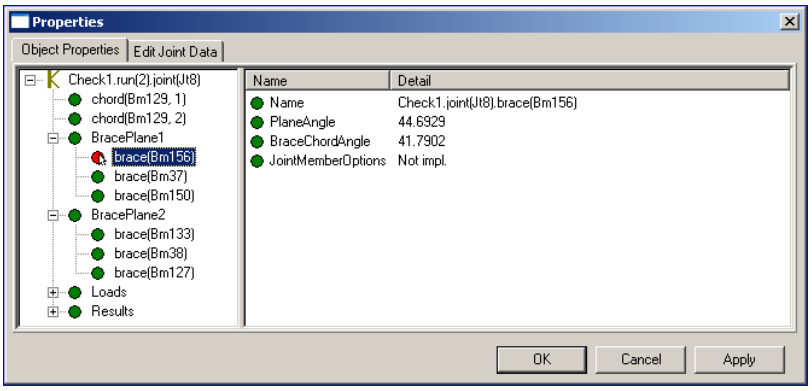
You may also modify member data for later use (see previous *Chapter 3.3.3 Define global joint parameters* and *Chapter 3.4.3 Modify local joint parameters).*

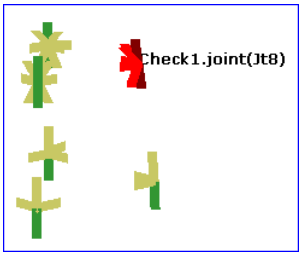


Member properties - options.

The Options folder for the Object Properties lists which members are part of the chord, how many planes are considered and the members that are considered braces for each plane.

Also notice that the plane angle and brace chord angle are also listed.

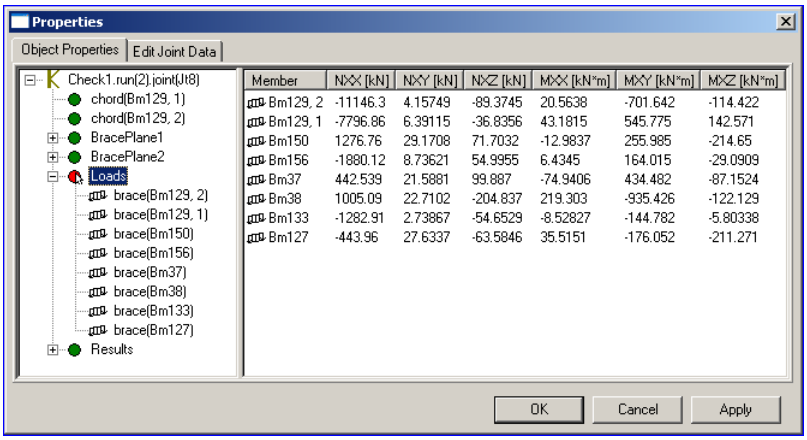




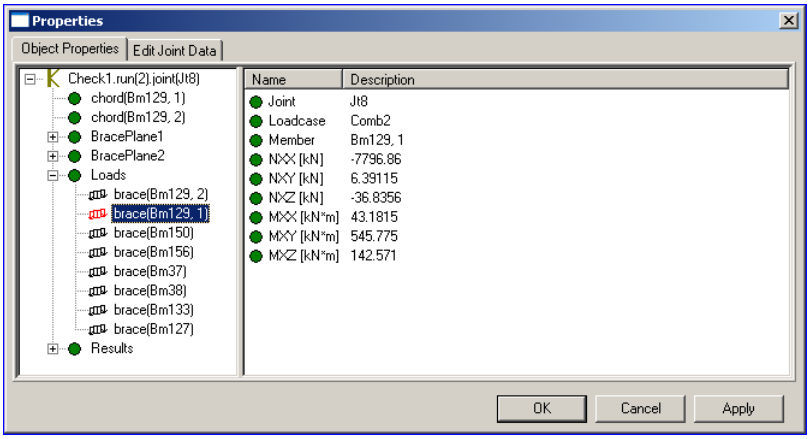
Member properties – code checking forces.

The code checking forces for each chord and brace member are listed as shown to the right.

These are computed at the joint end of the chord or brace.

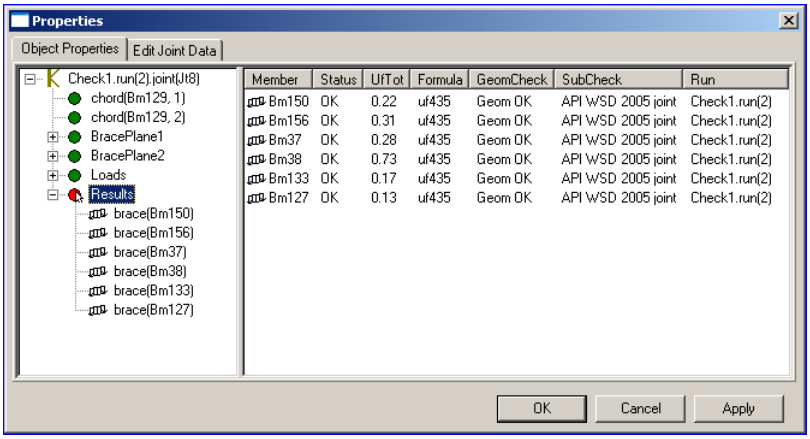


The code checking forces may also be shown per chord or brace.



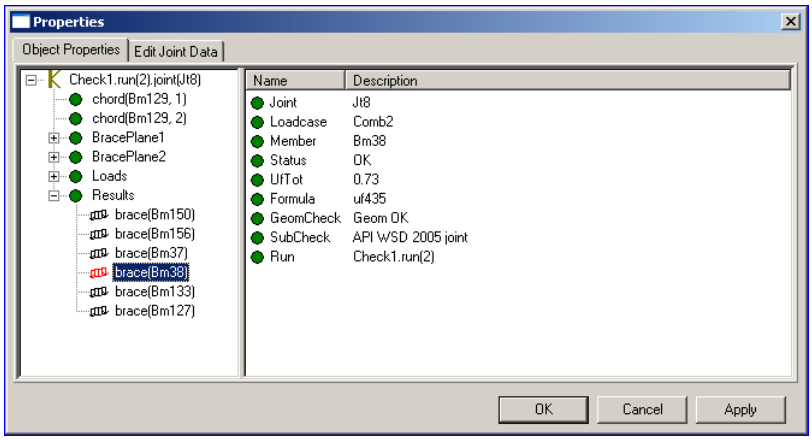
Member properties – code checking results.

An overview of code checking results for the joint is shown to the right.



For a full documentation of the nomenclature, please see *Appendix B Code checks and nomenclature*

The code checking results per brace may also be presented by selecting each individual brace.



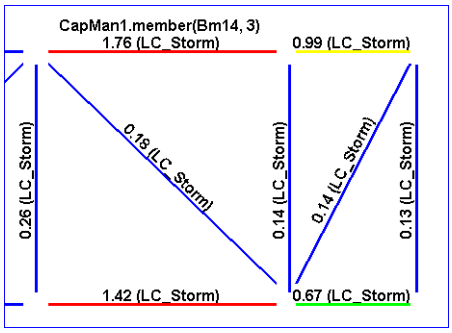
3.8 Modify and re-run code check

There are three levels of modifying a code check run

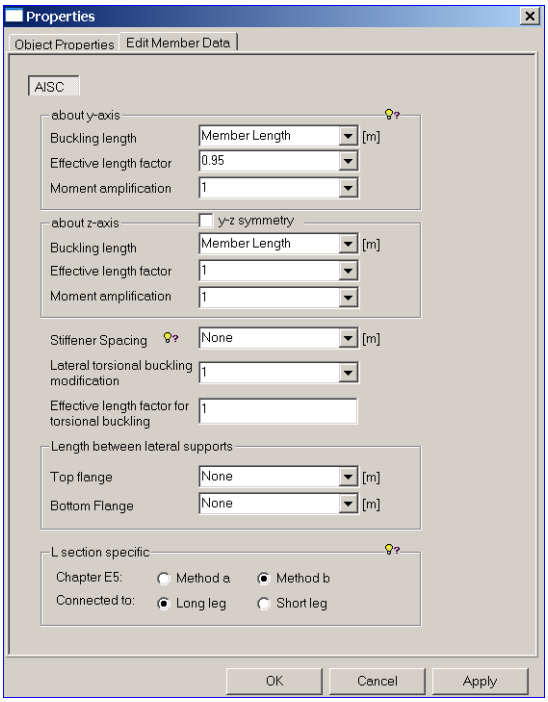
- Change code checking parameters like safety factors, buckling parameters and stiffener spacing. These are parameters that belong to the capacity model and a re-run of the code check does not require new code checking forces
- Change section or material properties of a beam without re-running the entire analysis. Please notice that the code checking result is based on a non-consistent stiffness or load matrix.
- Change section or material properties or adding/removing additional structural members using a consistent and updated load and stiffness matrix. This option requires a full re-run of the analysis and the code check steps.

Each of these options is described in the following using the following reference model:

Focus is member(Bm14,3).



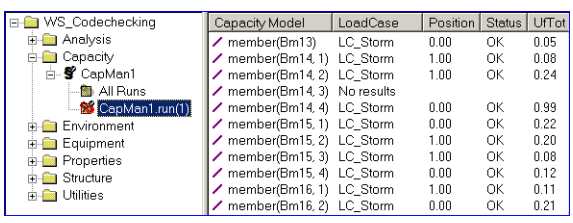
Name	Description
Member	Bm14, 3
Loadcase	LC_Storm
Position	0.00
Status	Failed(uf)
UfTot	1.76
Formula	ufH1
GeomCheck	Geom OK
SubCheck	AISC member
Run	CapMan1.run(1)
ufShear	0.220
ufH1	1.756
ufH1ax	0.007
ufH1mo	1.750
ufH2	0.000
ufH2ax	0.000
ufH2mo	0.000
ufH3	0.000
sldTens	0
sldComp	80.1709453



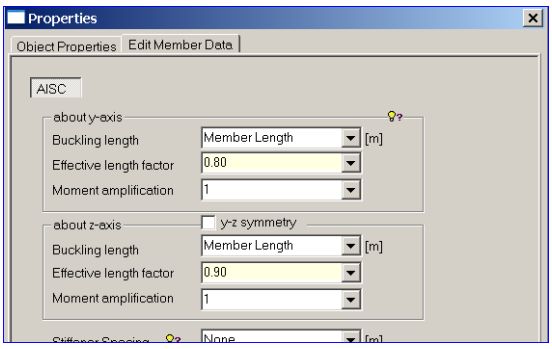
3.8.1 Change code checking parameters only

Modify the buckling lengths of member(Bm14,3) from browser or GUI (select the member, RMB and choose *Properties*). Change the effective length factors about local y and x axis to 0.80 and 0.90 respectively.

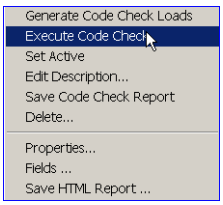
The browser is now updated since member(Bm14,3) has no results since the member data has been modified.,

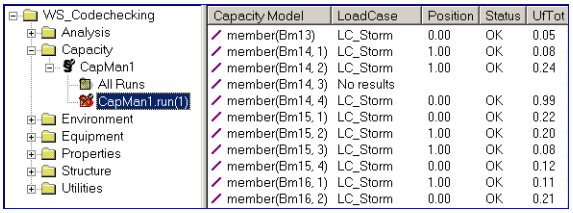


Capacity Model	LoadCase	Position	Status	UfTot
member(Bm13)	LC_Storm	0.00	OK	0.05
member(Bm14, 1)	LC_Storm	1.00	OK	0.08
member(Bm14, 2)	LC_Storm	1.00	OK	0.24
member(Bm14, 3)	No results			
member(Bm14, 4)	LC_Storm	0.00	OK	0.99
member(Bm15, 1)	LC_Storm	0.00	OK	0.22
member(Bm15, 2)	LC_Storm	1.00	OK	0.20
member(Bm15, 3)	LC_Storm	1.00	OK	0.08
member(Bm15, 4)	LC_Storm	0.00	OK	0.12
member(Bm16, 1)	LC_Storm	1.00	OK	0.11
member(Bm16, 2)	LC_Storm	0.00	OK	0.21



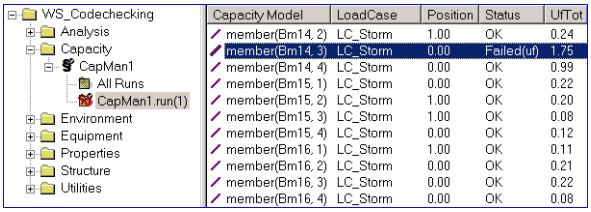
To generate new code check results you need to re-run the code check. Select the actual run, RMB and choose *Execute Code Check*.

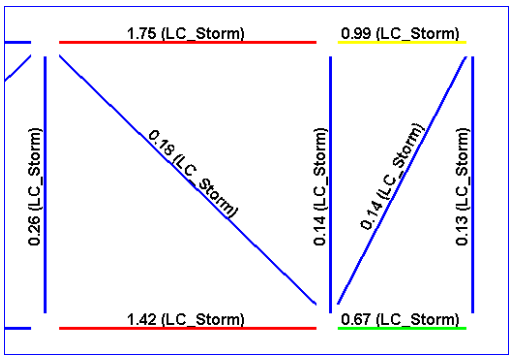




New results are now computed and may be investigated as explained in the previous Chapter.

As can be seen the utilisation factor has changed from 1.76 to 1.75.



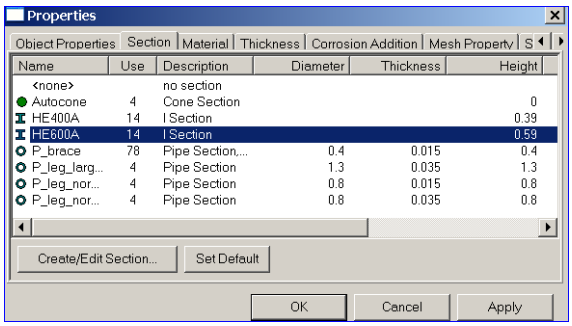


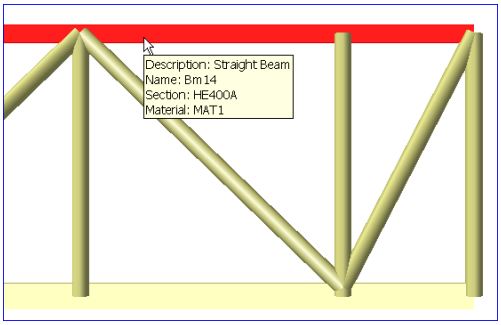
3.8.2 Modify structural data without re-running analysis

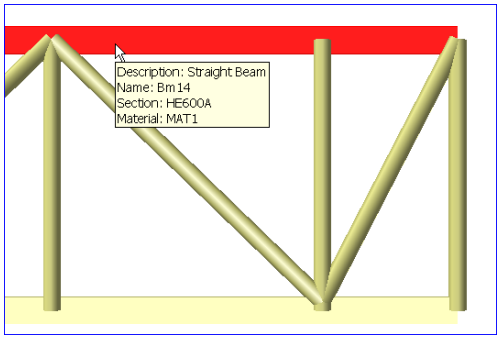
When you do modifications to the structural model (typically section or material data) without re-running the whole analysis it is required to re-generate the capacity members and code checking forces prior to executing the code check.

Notice also that if you have added local modifications to a capacity member these need to be re-applied. All global settings are kept.

The following example shows how to change the section type of a beam (from HE400A to HE600A) and compute a new utilisation factor without re-running analysis.

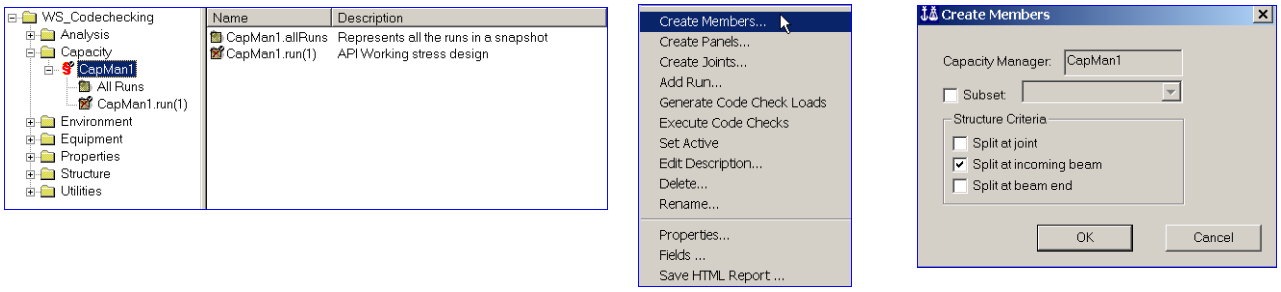

Properties
Object Properties | Section | Material | Thickness | Corrosion Addition | Mesh Property | S
Name | Use | Description | Diameter | Thickness | Height
<none> no section
Autocone 4 Cone Section 0
HE400A 14 I Section 0.39
HE600A 14 I Section 0.59
P_brace 78 Pipe Section... 0.4 0.015 0.4
P_leg_larg... 4 Pipe Section 1.3 0.035 1.3
P_leg_nor... 4 Pipe Section 0.8 0.015 0.8
P_leg_nor... 4 Pipe Section 0.8 0.035 0.8
Create/Edit Section...
Set Default
OK
Cancel
Apply





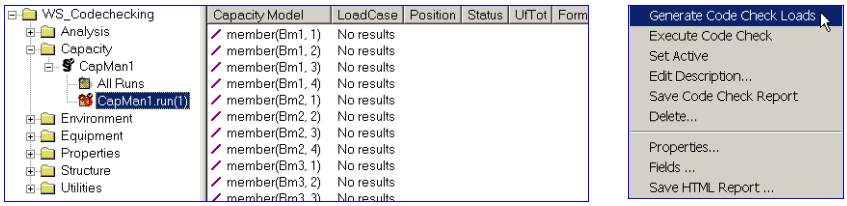
Create new members

Create new members from the same capacity manager (or make a new capacity manager). Select the capacity manager, RMB and choose *Create Members*. Use the same split option as in the previous run – in this case "Split at incoming beam".



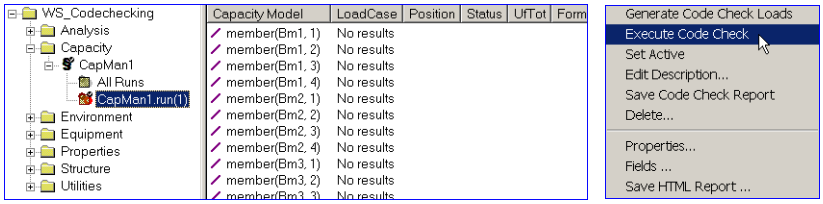
Generate new code checking forces

From the browser, right click to re-create the code checking forces



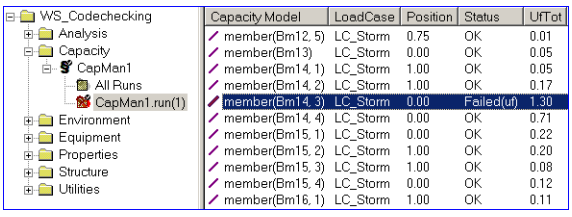
Execute the code check

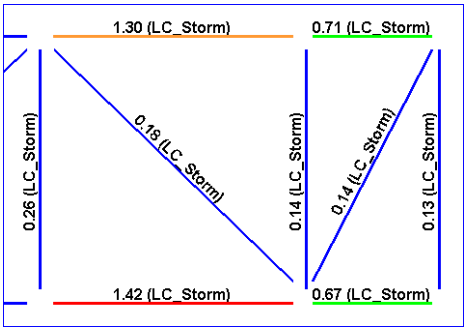
Start the code check from the browser (select the actual run, RMB and *Execute Code Check*).



As can be seen the utilisation factor drops from 1.76 to 1.30 without re-running structural analysis.

- The global member settings are the same as before, any local modifications to the member must be re-applied when creating new members
- The revised code check is based on previous finite element analysis where Bm14 has section type HE400A, the code check however uses HE600A during the check





3.8.3 Modify structural data and re-run analysis

When you re-run analysis it is necessary to

- Create new members of the capacity manager.
    - Global settings are kept, but local modifications must be re-applied
- Compute new code checking forces
- Run the code check

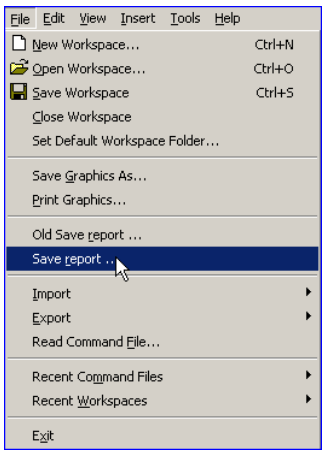
To do this you run analysis (ALT+D) and follow the steps as explained in the previous Section. New results may now be assessed; typically:

- Modification of section profile for Bm14 from HE400A to HE600A without re-running analysis gives an utilisation factor 1.30.
- For the same change and re-running the analysis, the utilisation factor becomes 1.28.

Notice that GeniE will check whether the concept model is more recent than the FE-analysis and give an error message when such situations are detected.

3.9 Make a report

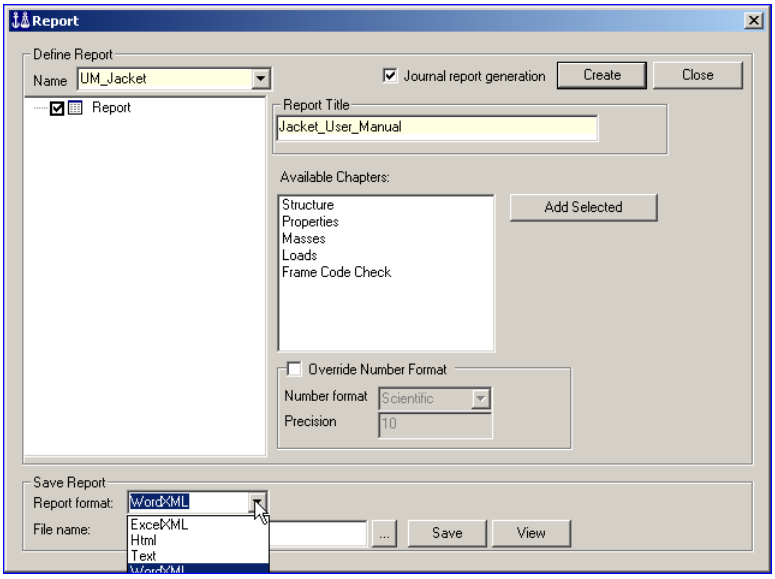
A customised report may be generated from the ***File|Save report***. Please notice that the report functionality in versions prior to GeniE version v3.4-27 is still available from the command ***File|Old Save report***.



The **File|Save report** allows you to specify the content of your report as well as to decide the levels of detail in of your report.

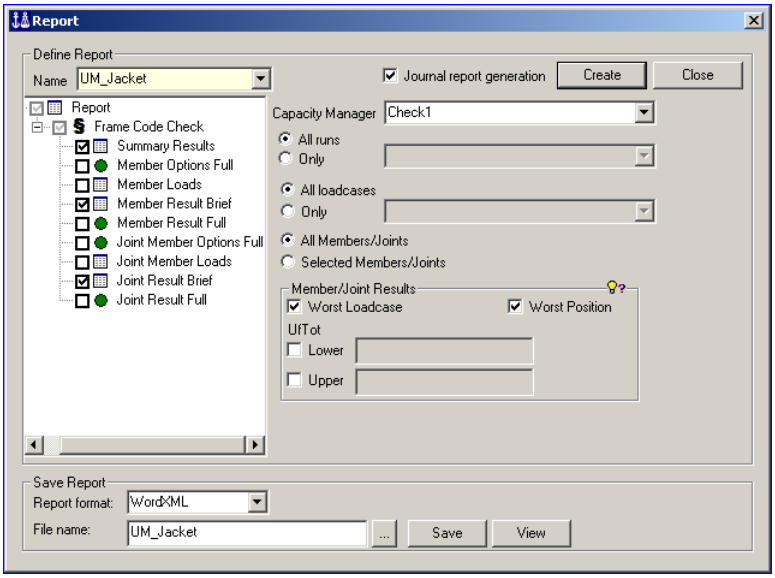
In the following is given an example on how to make a code checking report.

In this case the report UM_Jacket will be made using the XML format for MS Word. You may also generate a report based for viewing in Excel, Html and a pure text based file. The commands used to generate the report are scripted (i.e. the same report is part of the journal file).



When you select Frame Code Check from the available chapters above (remember to click “Add Selected) you can decide the granularity of your report.

Notice that if you want to use the option “Selected Members/Joints” these must be the current selection (from the browser or the GUI) before you create the report.



This page is blank for printing

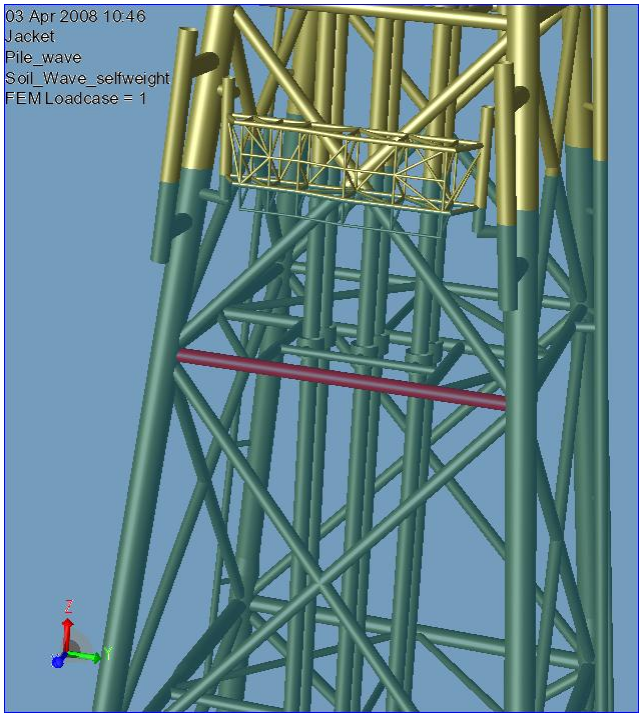
4. BEAM STRESSES

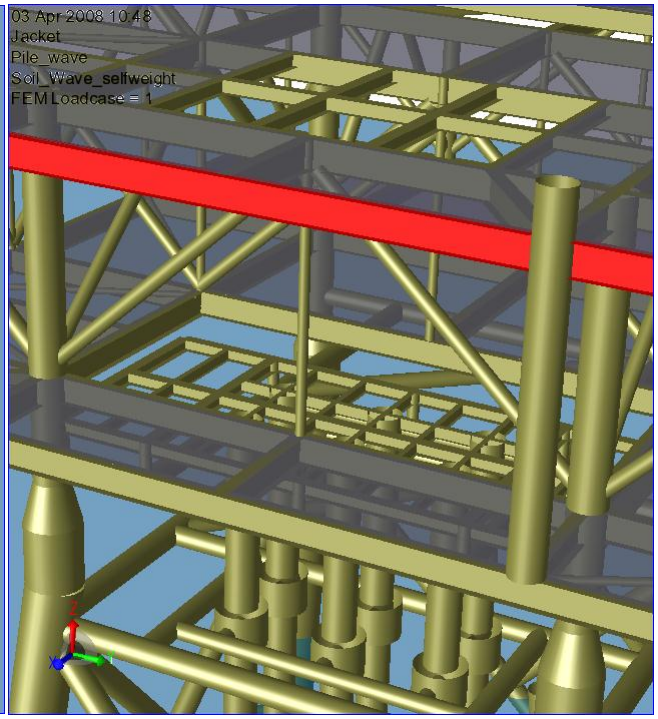
This Chapter explains how to present and export beam force & stress data to Excel or other tools supporting the xml standard. The procedure is as follows:

- Select a beam or continuous beams
- Activate the 2D force/beam presentation tool from menu ***Tools|Analysis|Beam Result Diagram***
- Select which force or stress components, loadcases and hotspots (or stress points)
- Display the graph(s) and add labels as desired – data can be exported via xml to e.g. Excel
- Look at print table for maximum and minimum value
- Look at detailed print containing all information

Stresses and forces will be presented for the so-called hotspots for positions along the beam. There are 5 predefined positions (start, 25%, 50%, 75% and end) plus those computed when there are variations in properties (like in a segmented member) and local moments outside the previous mentioned positions (see also Section 3.5 Code checking positions and forces).

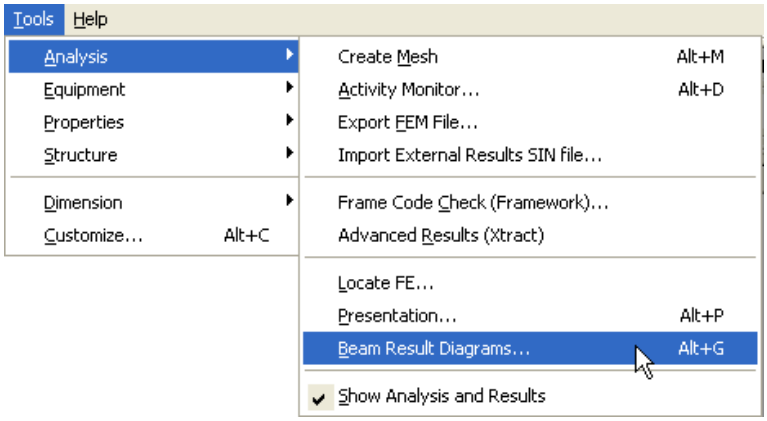
Two models will be used to show results; a tubular member in a jacket and an I-profile part of a topside. Prior to presenting beam forces and moments, result must be available from an analysis.





Notice that it is also possible to present beam forces in a 3D view using the options from ***Tools|Analysis|Presentation***. This will give you an overview of forces in the whole model or parts of it – for details, the diagram tool should be used.

The beam result diagram tool is started from ***Tools|Analysis|Beam Result Diagram***.



Some limitations apply regarding which components that can be presented for different types of structure in the beam result diagram. An overview of this is displayed below.

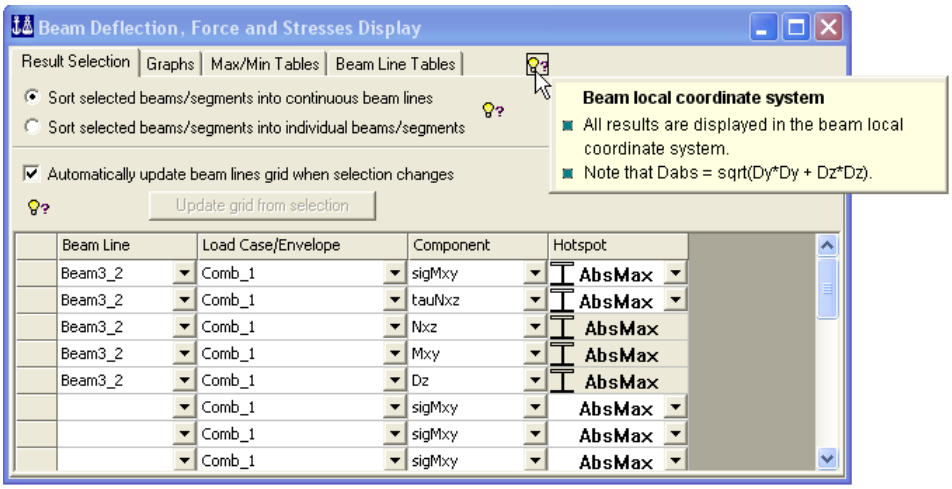
Beam Result Diagram - Components supported for different structure

Curve type	**Element type**	**Components**	**Parameterized cross sections** I,L,Pipe,Bar, etc.	**General Cross Section**
Straight Beam	1st order	Force	Y	N
Pile	1st order	Force	Y	N/A
Curved Beam	1st order	Force	N	N
Straight Beam	2nd order	Force	Y	N
Pile	2nd order	Force	Y	N/A
Curved Beam	2nd order	Force	N	N
Straight Beam	1st order	Stress	Y	N
Pile	1st order	Stress	Y	N/A
Curved Beam	1st order	Stress	N	N
Straight Beam	2nd order	Stress	Y	N
Pile	2nd order	Stress	Y	N/A
Curved Beam	2nd order	Stress	N	N
Straight Beam	1st order	Deflection	Y	Y
Pile	1st order	Deflection	Y	N/A
Curved Beam	1st order	Deflection	N	N
Straight Beam	2nd order	Deflection	Y	Y
Pile	2nd order	Deflection	Y	N/A
Curved Beam	2nd order	Deflection	N	N

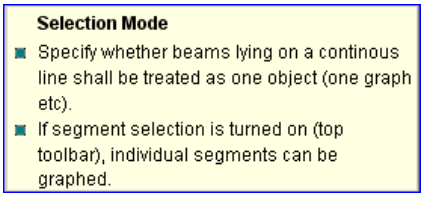
Note: Only pipe-sections can be used for piles.

4.1 Result Selection

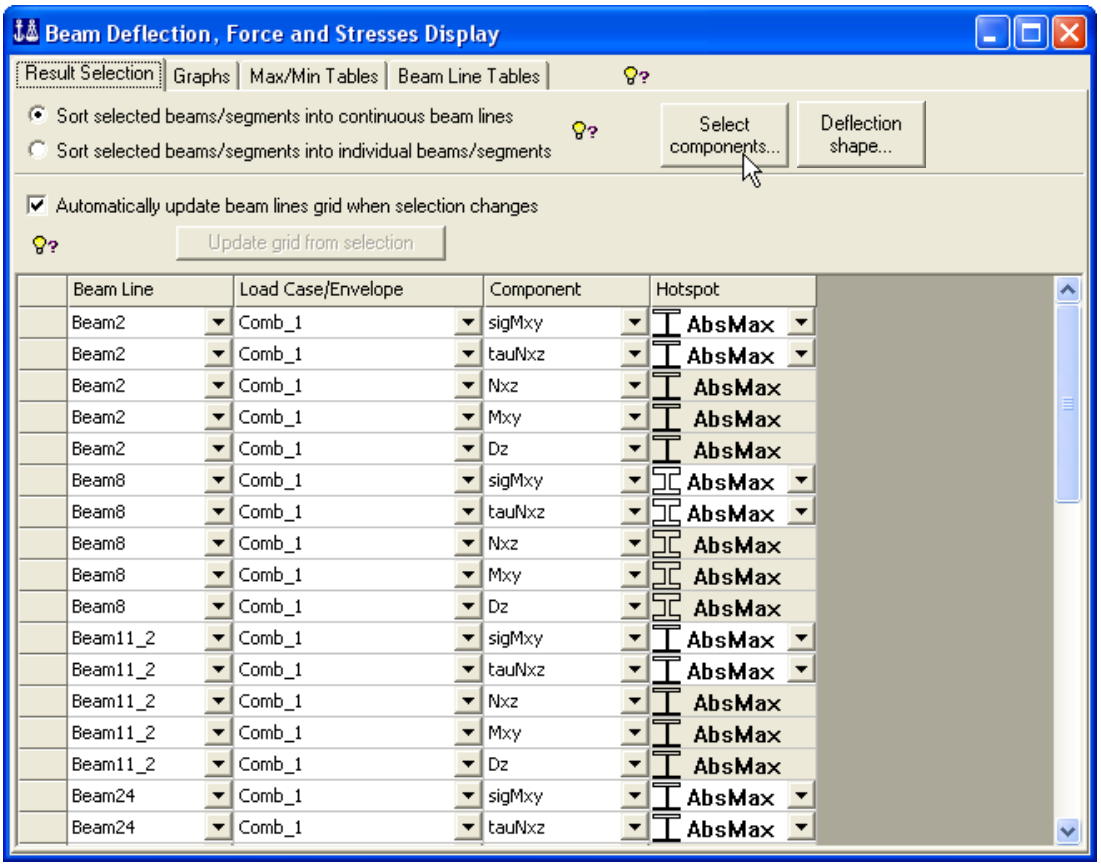
Per default the diagram tool will present the force and stress components for shear and moment (tauNxz, sigMxy, Nxz, Mxy, Dz).

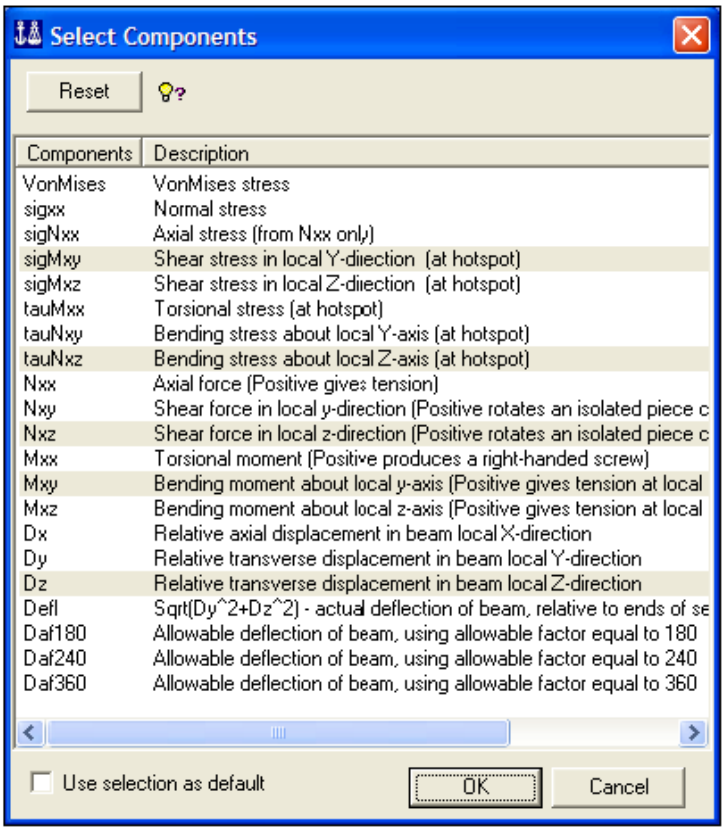

Beam Deflection, Force and Stresses Display
Result Selection
Graphs
Max/Min Tables
Beam Line Tables
Sort selected beams/segments into continuous beam lines
Sort selected beams/segments into individual beams/segments
Automatically update beam lines grid when selection changes
Update grid from selection
Beam local coordinate system
All results are displayed in the beam local coordinate system.
Note that Dabs = sqrt(Dy*Dy + Dz*Dz).
Beam Line
Load Case/Envelope
Component
Hotspot
Beam3_2 Comb_1 sigMxy AbsMax
Beam3_2 Comb_1 tauNxz AbsMax
Beam3_2 Comb_1 Nxz AbsMax
Beam3_2 Comb_1 Mxy AbsMax
Beam3_2 Comb_1 Dz AbsMax
Comb_1 sigMxy AbsMax
Comb_1 sigMxy AbsMax
Comb_1 sigMxy AbsMax

In case you have selected several beams you may present results either as a continuous line or into individual lines.


Selection Mode
Specify whether beams lying on a continous line shall be treated as one object (one graph etc).
If segment selection is turned on (top toolbar), individual segments can be graphed.

By clicking the button “Select Components” you open the dialog “Select Components”.





In the "Select Components" dialog you can easily select the components of interest for all your selected beams. Click on a component to select it. Use shift+click to select several components.

If you tick the checkbox "Use selection as default" the current selection of components will be stored as the default setting. Your default selection is stored even if you exit GeniE.

To reset the selection to the GeniE default, click the "Reset" button.

Note on Allowable Deflection of Beam:

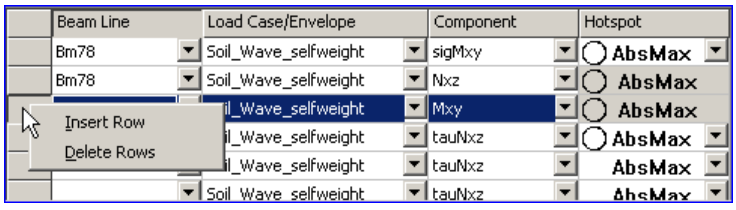
The check is as follows: Actual deflection < Allowable deflection

Actual deflection is the deflection as computed by GeniE.

The total length from the start to the end of the beam diagram is used. Intermediate points are not considered. It is important that you make sure you are only selecting the span between 2 supports when you are validating that the deflections are smaller than allowable deflection.

There are a number of options to select to present various attributes. The on-line help (click on the light-bulb) explains how you may select beams, load cases, component and hotspots. Some examples are also listed in the following.

You can also delete and add rows by RMB on the left mouse column as shown below.

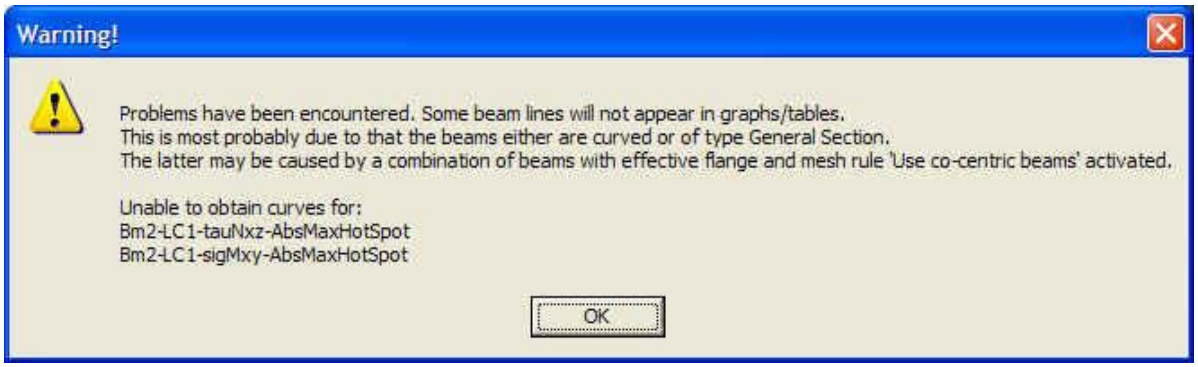


Selection Grid

- In the grid below you may specify which loading condition(s), result component and hotspot that shall be graphed/tabulated.
- The beam lines are taken from the current selection. You may change the selection while this dialog is running to view different beams.
- When the selection changes new beam lines will receive the same settings as the beam lines currently in the table.
- The same beam line may be selected several times for comparison of different results.
- By selecting multiple rows in a column and changing the combo selection on the last row, multiple rows can be edited.
- To remove rows, press the left most column in a row to select the entire row and press the "Delete" button.

4.1.1 Limitation on stress computation

If an 'Effective Flange' property is assigned to a beam and the mesh setting 'Use co-centric beams' is checked, stresses will not be computed for that beam neither in the 'Beam Result Diagram' tool nor in the 'Report->FEM Results'. The 'Beam Result Diagram' tool will give the user a warning in a separate dialog.


Warning!
Problems have been encountered. Some beam lines will not appear in graphs/tables.
This is most probably due to that the beams either are curved or of type General Section.
The latter may be caused by a combination of beams with effective flange and mesh rule 'Use co-centric beams' activated.
Unable to obtain curves for:
Bm2-LC1-tauNxz-AbsMaxHotSpot
Bm2-LC1-sigMxy-AbsMaxHotSpot
OK

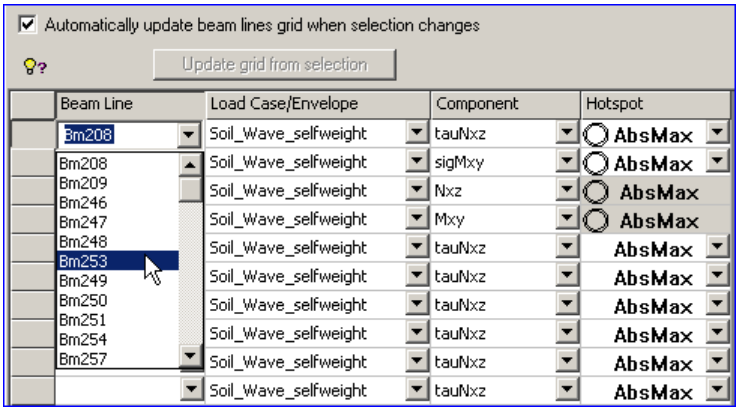
In the report there will be no results for this beam.

LC1 : FEM Beam Stress SUB PAGE: 1

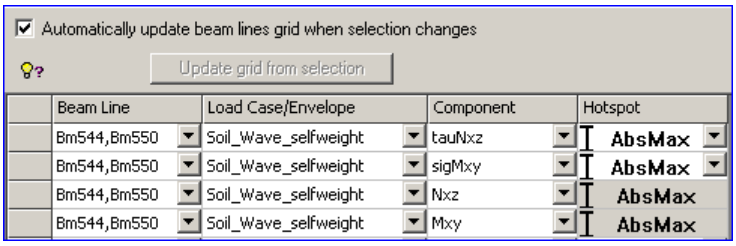
Name	Position	Length [m]	Algo	Compo	Hotspot SigMxz [Pa]	VonMises [P TauMxx [Pa]	Sigxx [Pa] TauNxy [Pa]	SigNxx [Pa] TauNxz [Pa]	SigMxy [Pa] Comment
Bm1									
Bm2									
Bm3									
Bm4									
Bm5									

4.1.2 Select beams

You may alter your selection from a previously graphic selection. Assume you have made a multiple selection you can use the combo selection to decide which beam to use when presenting attributes. In this case the next presentation will yield Bm253.



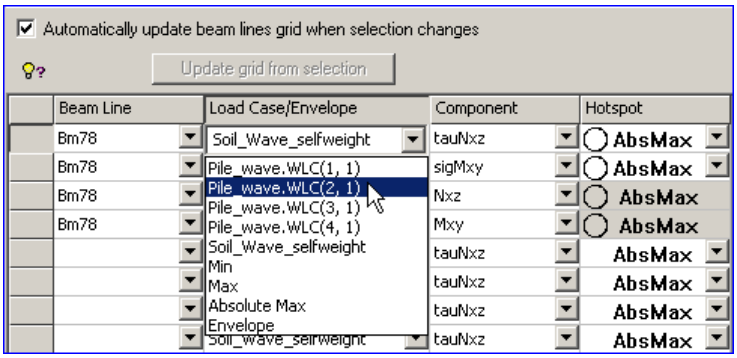
If you have selected two continuous beams, they will appear in the combo box as follows:



4.1.3 Select loadcases

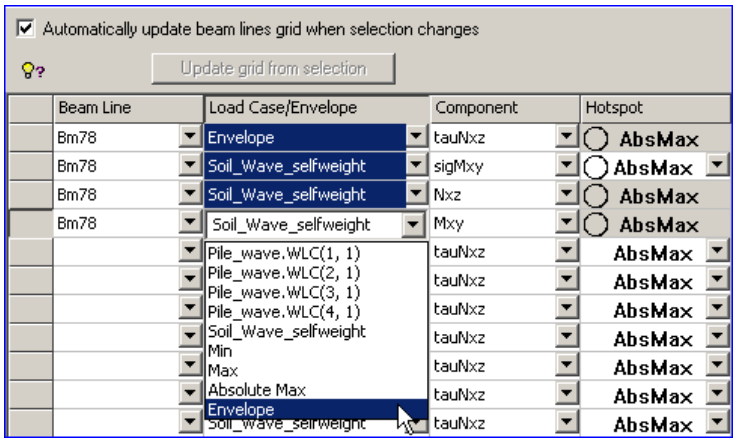
It is possible to present results for individual loadcases and minimum, maximum, absolute maximum and an envelope for a set of loadcases. These are all selected from the combo box.

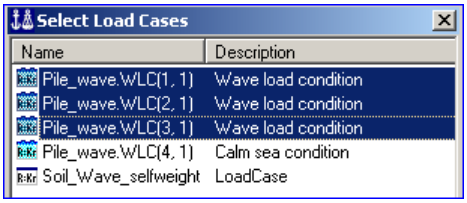
The example to the right shows that the wave loadcase WLC(2,1) will be used.



For the other options you need to decide which of the loadcases that shall be part of the selection.

The picture to the right shows that an envelope shall be presented for all the selected components. The below dialogue will appear and in this case three loadcases have been selected to be part of the envelope.

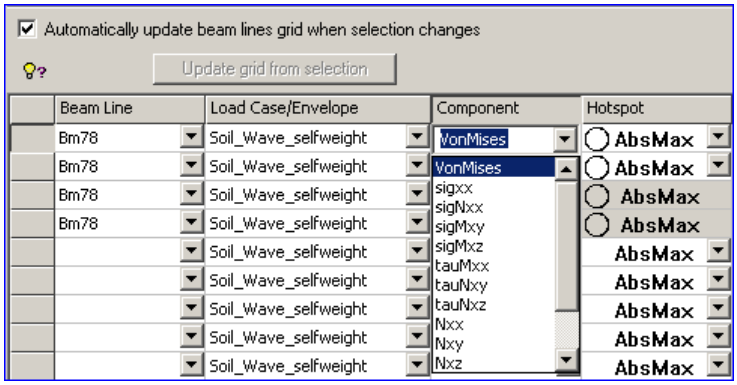




4.1.4 Select component

The beam result diagram tool will present results for all force, deflection and stress components including VonMises stresses.

In the example to the right, VonMises stress is selected.



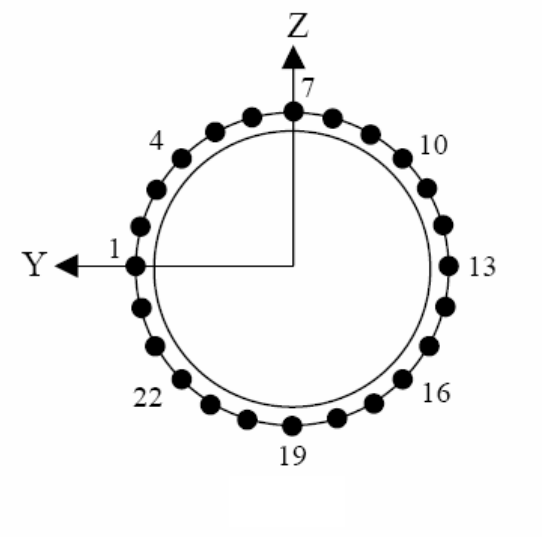
The stress components are:

- vonMises: Equivalent stresses according to the VonMises criteria
- sigxx: Combined axial stress from axial force and bending moments
- sigNxx: Normal stress due to axial force alone (Nxx)
- tauNxy: Shear stress due to shear force in y direction (Nxy)
- tauNxz: Shear stress due to shear force in z direction (Nxz)
- tauMxx: Shear stress due to torsional moment (Mxx)
- sigMxy: Normal stress due to bending moment about y-axis (Mxy)
- sigMxz: Normal stress due to bending moment about z-axis (Mxz)

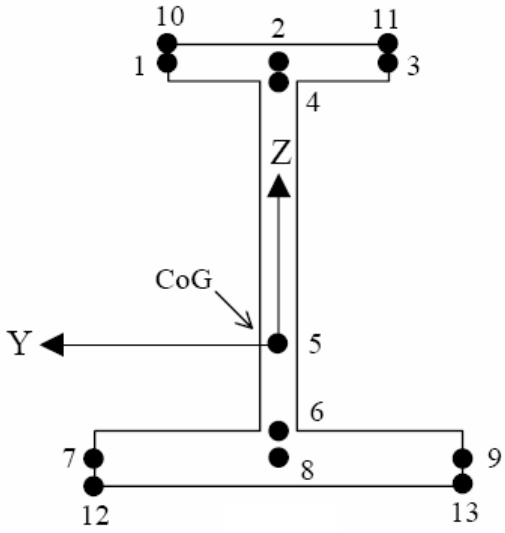
4.1.5 Select hotspot position

The hotspots (or stress points) vary from profile to profile. In the following the hotspot numbering is shown for the relevant profile types. Selecting a hotspot will report values for this hotspot only. Selecting e.g. max and min will report for such positions only; this means that the hotspot reported may vary along the beam length.

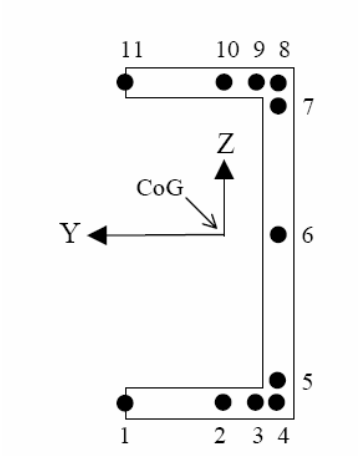
Pipe:



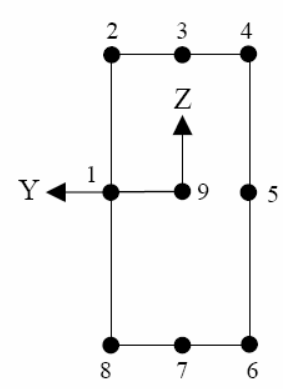
I or H-profile:



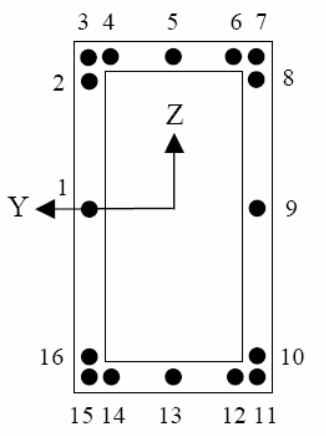
Channel:



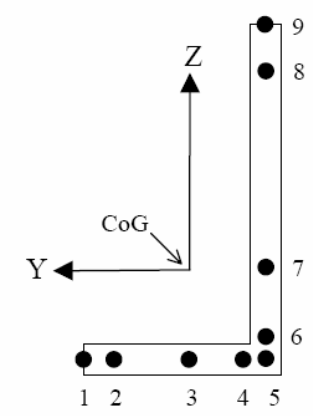
Bar:



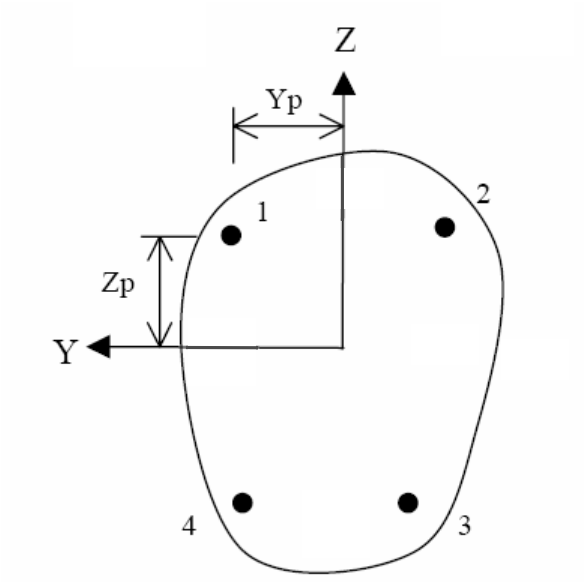
Box:



L-profile:



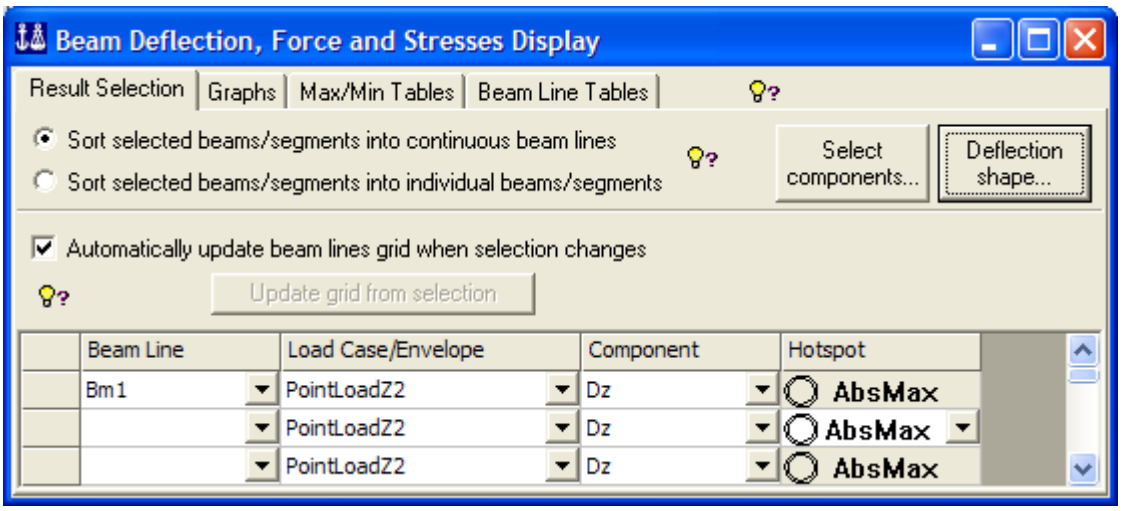
General section:



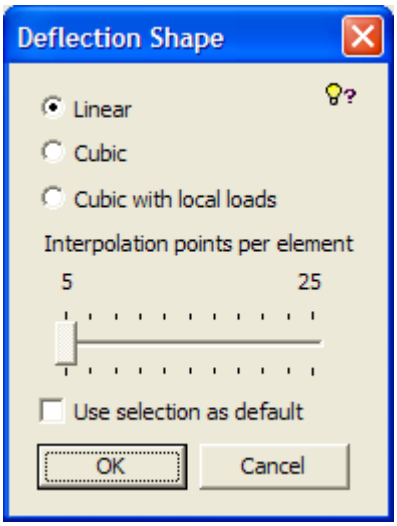
The positions are calculated as follows:

±Iy/Wy and ±Iz/Wz.

4.1.6 Deflection shape



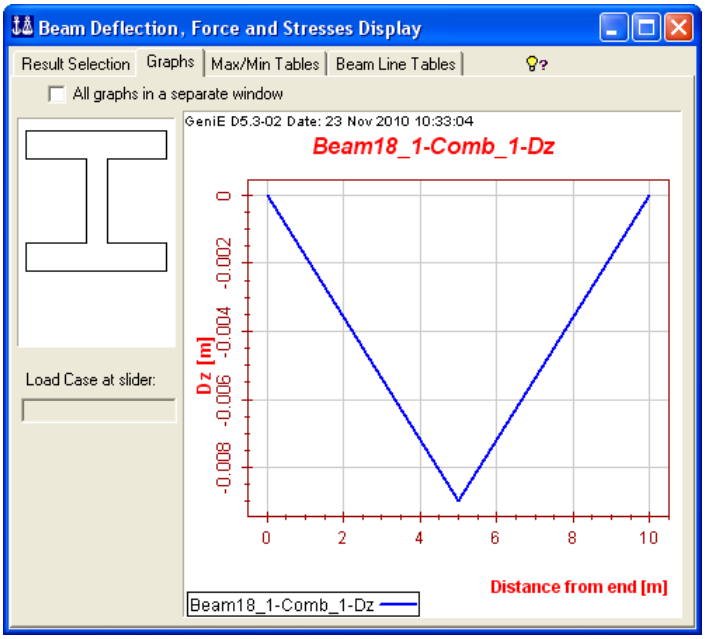
By clicking the button "Deflection shape" you get to select which method you want to use to present the graph.



There is also a slider to select number of interpolation points per element. A higher number of interpolation points will produce a smoother looking graph, but it requires slightly more time for drawing.

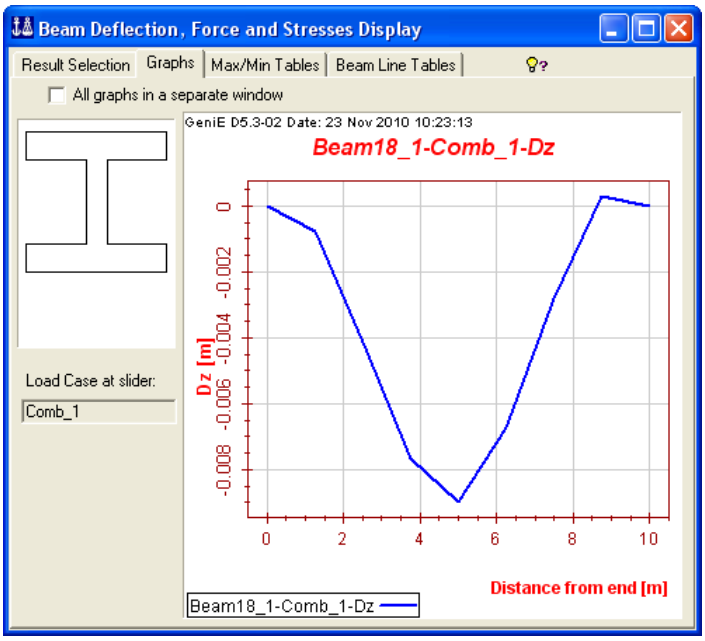
The interpolation points are only relevant for the two cubic options.

Linear
This implies that each finite element is approximated by a straight line.

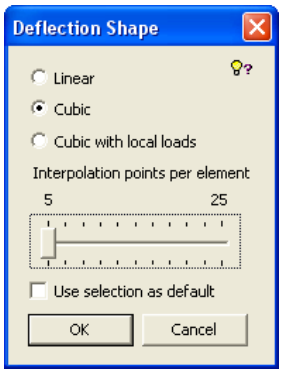


Cubic

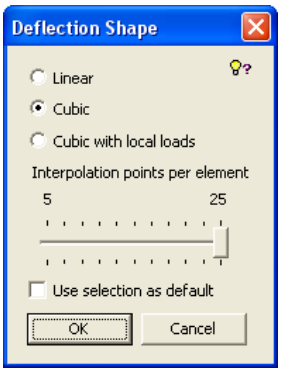
This means that each finite element is approximated by a cubic spline function that matches the beam end rotations. Representation is always linear, but interpolation points are calculated using cubic shape functions.

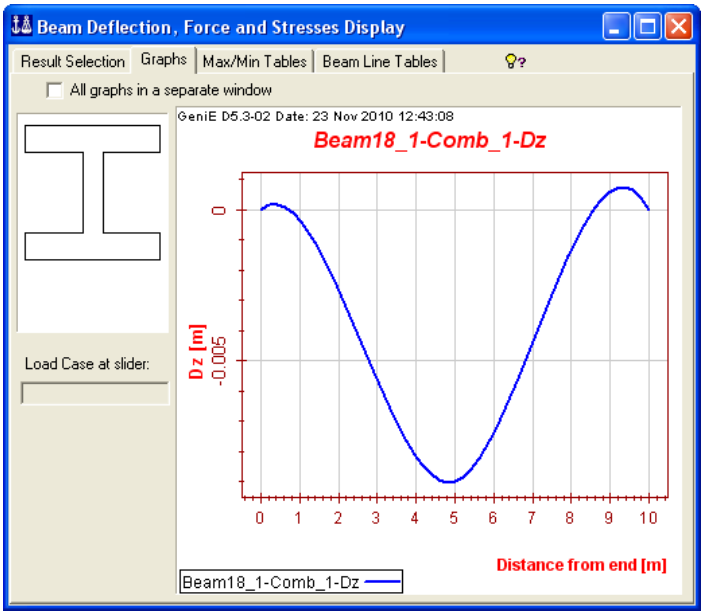


The graph in the illustration to the right is drawn with 5 interpolation points per element.



The graph in the illustration to the right is drawn with 25 interpolation points per element, note the difference in smoothness between the two graphs

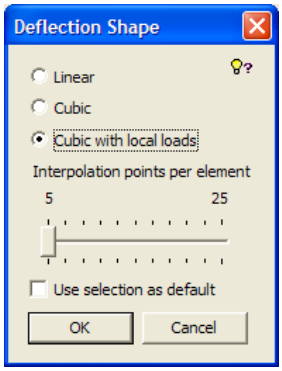


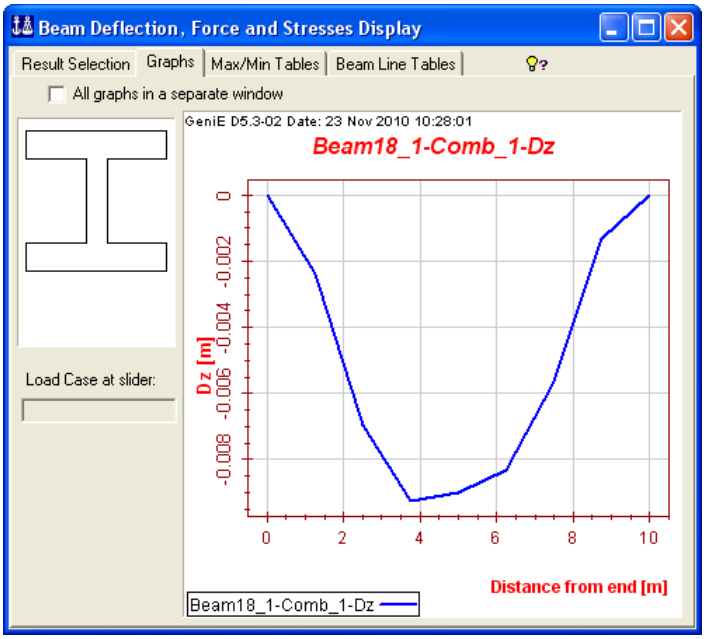


Cubic with local loads

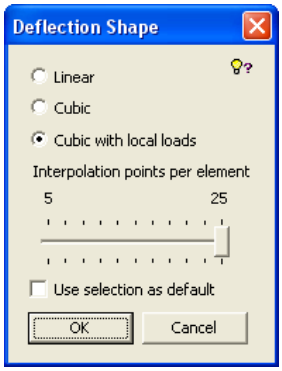
An analytic solution of deflection due to local element loads is superimposed on the cubic spline function

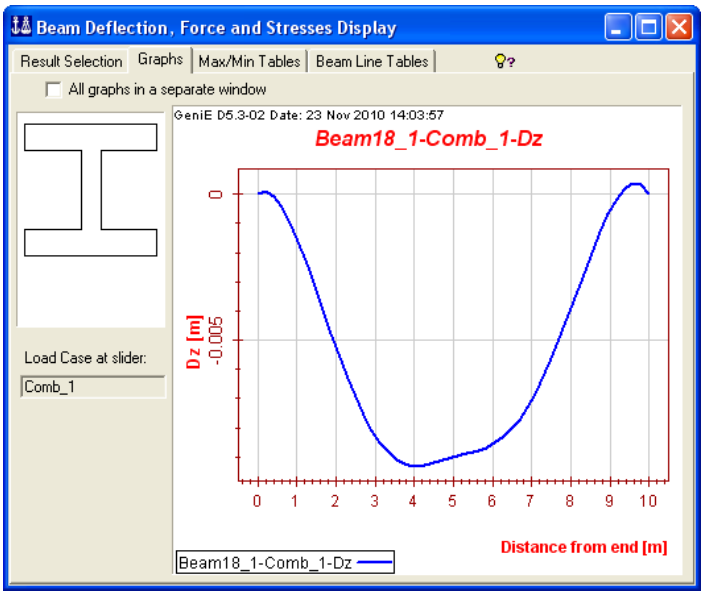
The graph in the illustration to the right is drawn with 5 interpolation points per element.





The graph in the illustration to the right is drawn with 25 interpolation points per element,





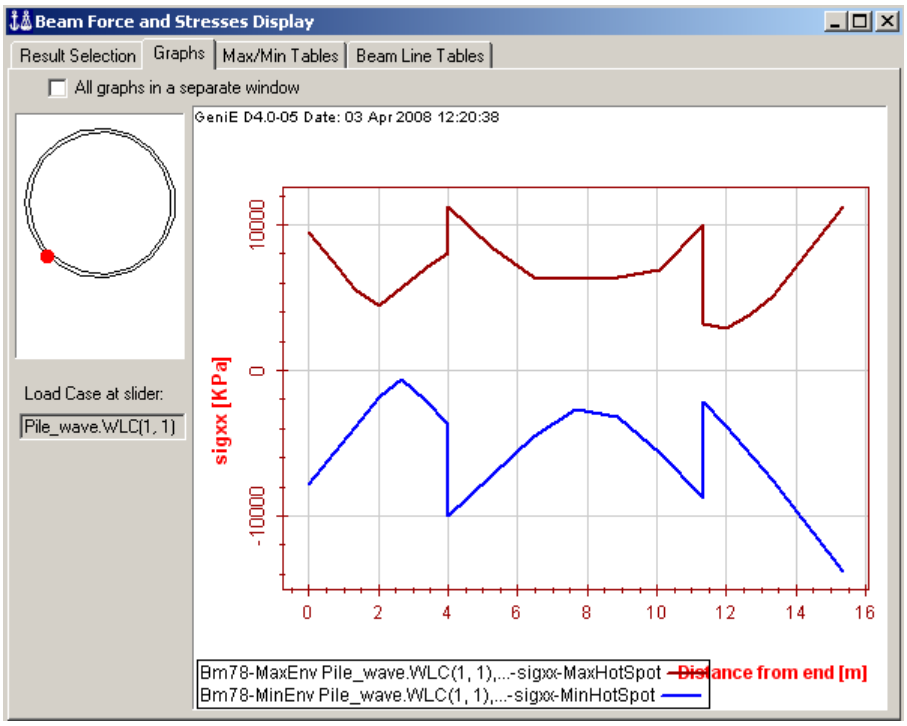
4.2 Graphs

A graph (or graphs) is calculated by clicking on the tab “Graphs”.

In this case one single beam has been selected. The component sigxx is shown for an envelope consisting of three wave load cases.

In the following some examples are given on how to label results, how to extract information and how to change font sizes for better picture generation.

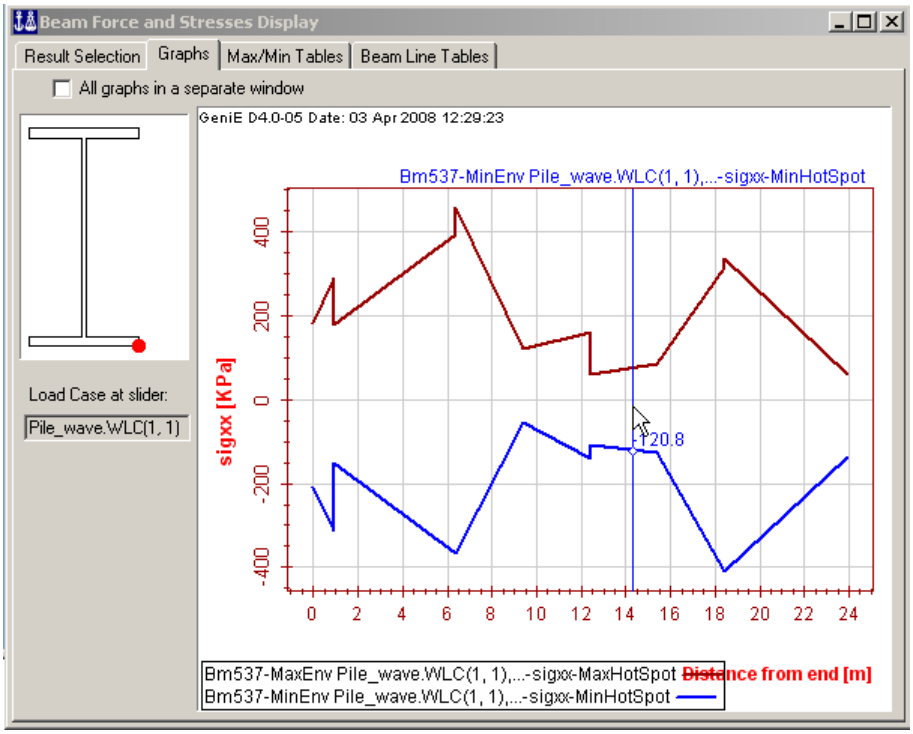
The red “dot” indicates the worst position (hotspot) that is shown.



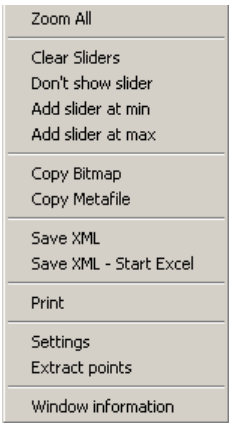
When showing results for an I-profile the graph looks different since the hotspot positions are different.

By moving the mouse along the x-axis, the actual value is shown and if the hotspot position changes, this is reflected in the left figure.

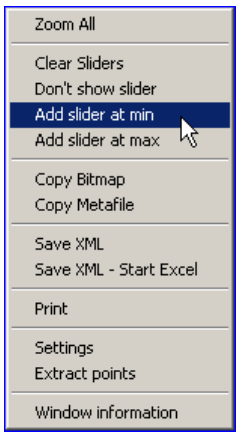
You switch curve to present by moving the mouse over the desired curve and click **CTRL+LMB**.

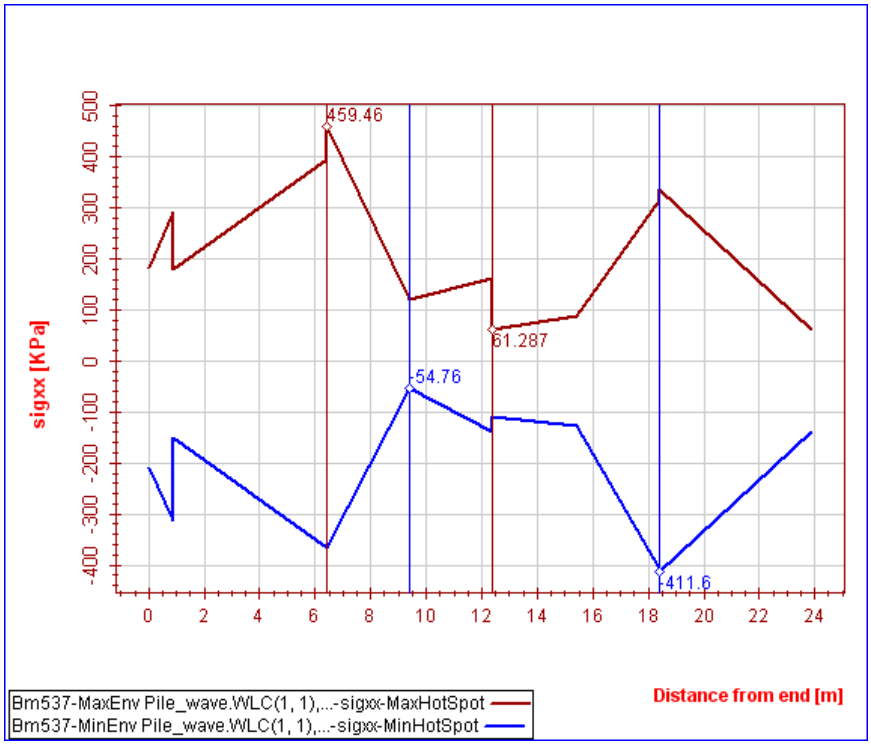


Right click in the graph window gives you access to the relevant commands.

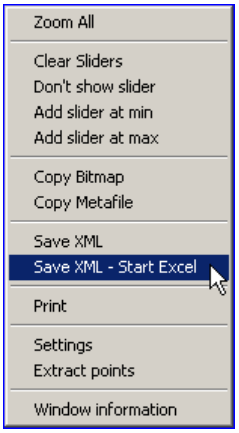


Labelling maximum and minimum values might be done in one operation by selecting "Add slider at min" and "Add slider at max".



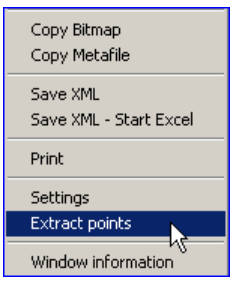


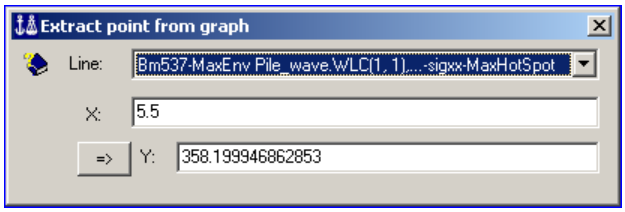
The graph values may also be exported to e.g. Excel using the xml format.

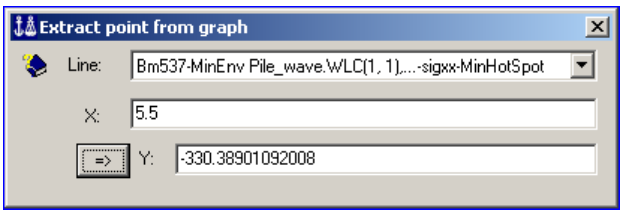


	A	B	C
1	Distance from end [m]	Bm537-MinEnv Pile_wave.WLC(1, 1),...-sigxx-MinHotSpot	Bm537-MaxEnv Pile_wave.WLC(1, 1),...-sigxx-MaxHotSpot
2	0.000239002	-209.956	182.605
3	0.899761	-314.082	290.764
4	0.900239	-149.978	177.789
5	6.39976	-365.679	393.49
6	6.40024	-365.567	459.469
7	9.4	-54.7692	121.781
8	12.3998	-139.203	161.218
9	12.4002	-109.839	61.2874
10	15.4001	-127.067	86.9818
11	18.4	-408.336	313.909
12	18.4005	-411.682	336.178
13	23.9	-137.437	61.9329

You may also find the value at a given position. This example shows how to find the value at position 5.5 m (this value is manually specified in the dialogue box as shown):

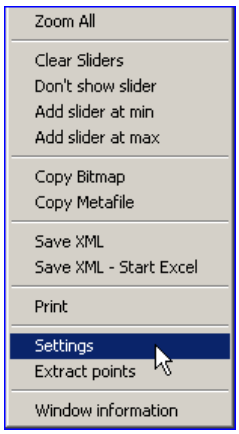


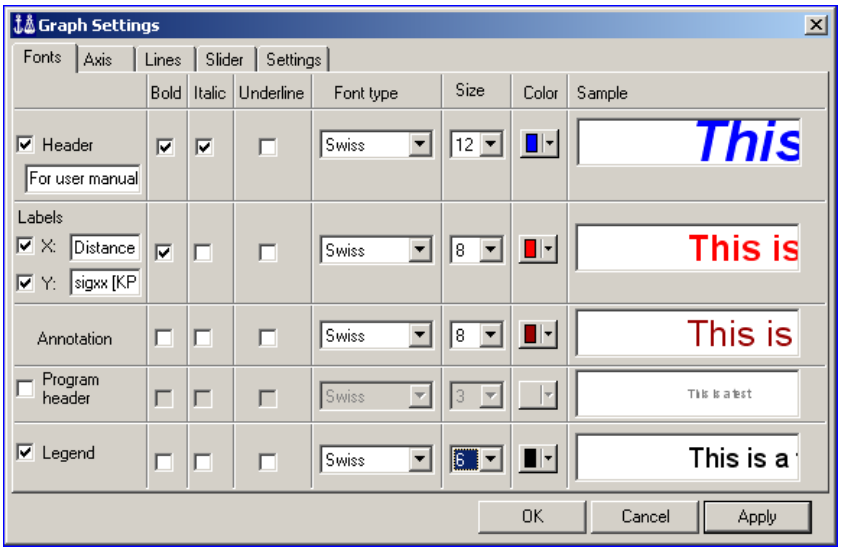




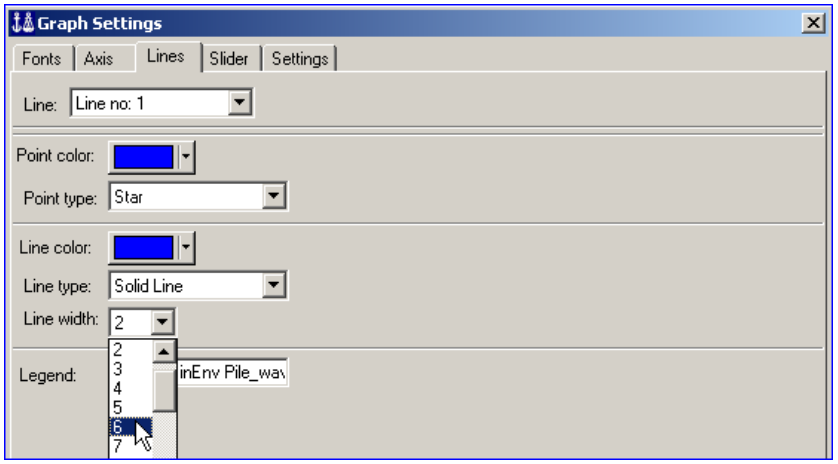
Each graph may be customised in terms of fonts and colour appearance. In this example the font size has been changed as well as the colour and thickness of a graph line

Adding header and change fonts:

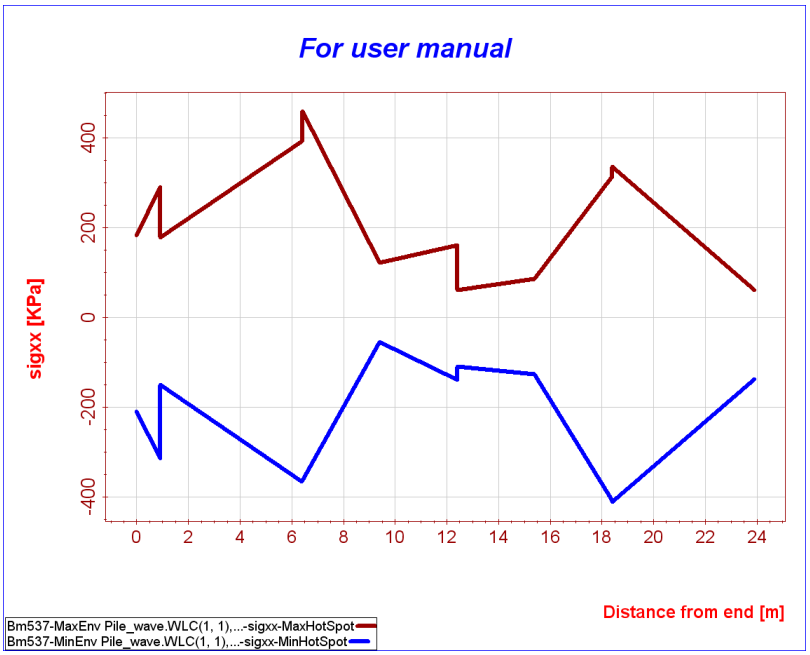




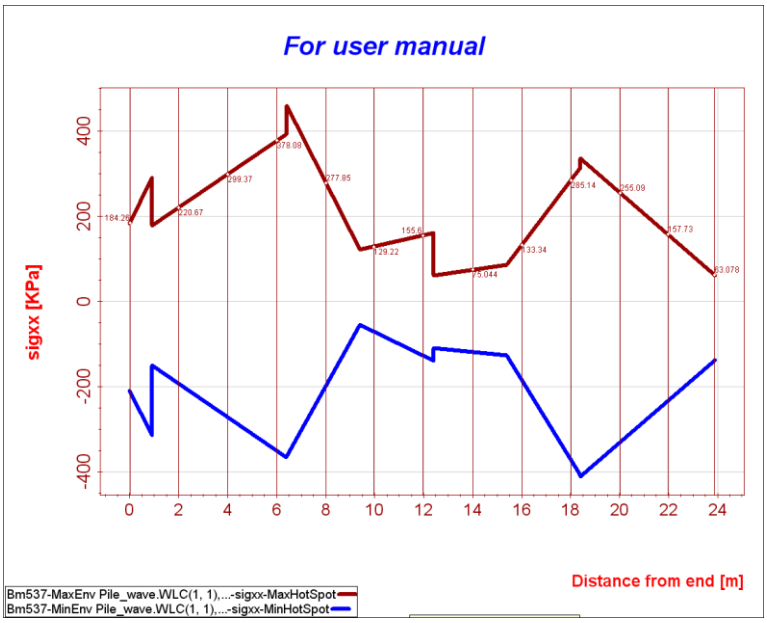
Modifying graph lines:



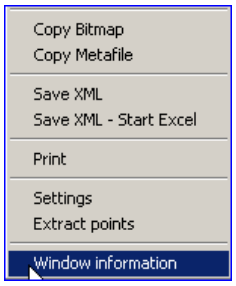
The new graphs look like:

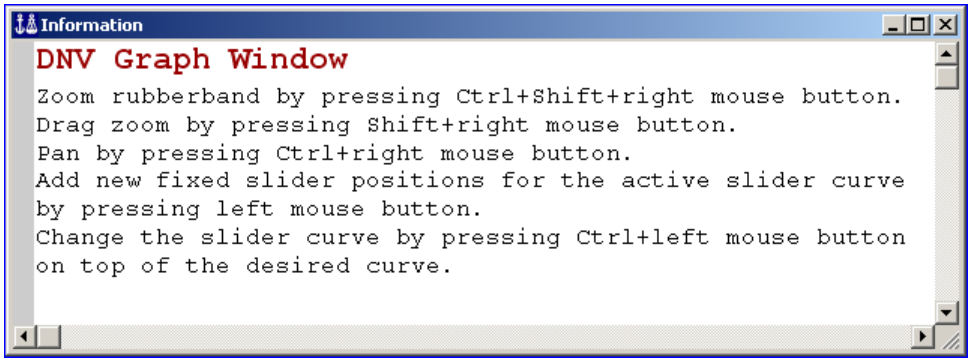


Labelling at specific positions is done by placing the cursor at the desired positions and click LMB. This example shows that labels have been added at each 2nd meter.



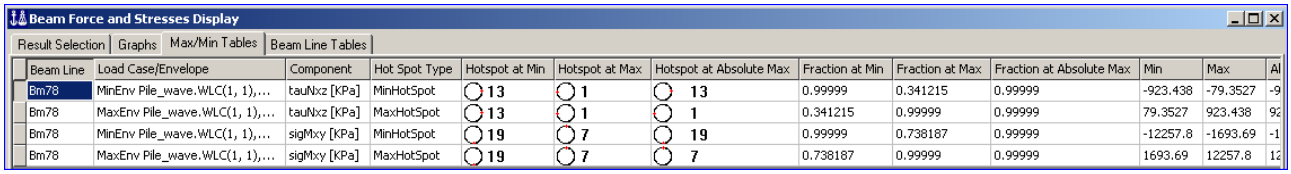
More information on how to modify the graphs and its settings may be found from:




Information
DNV Graph Window
Zoom rubberband by pressing Ctrl+Shift+right mouse button.
Drag zoom by pressing Shift+right mouse button.
Pan by pressing Ctrl+right mouse button.
Add new fixed slider positions for the active slider curve by pressing left mouse button.
Change the slider curve by pressing Ctrl+left mouse button on top of the desired curve.

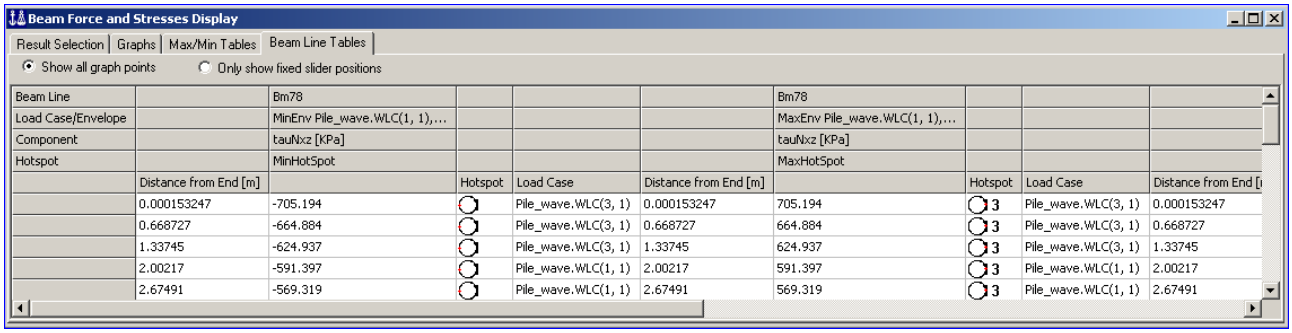
4.3 Maximum and detailed values

Maximum and minimum values are presented by using the tab "Max/Min table". This report gives the relevant peak values.



Beam Line	Load Case/Envelope	Component	Hot Spot Type	Hotspot at Min	Hotspot at Max	Hotspot at Absolute Max	Fraction at Min	Fraction at Max	Fraction at Absolute Max	Min	Max	Al
Bm78	MinEnv Pile_wave.WLC(1, 1),...	tauNxz [KPa]	MinHotSpot	13	1	13	0.99999	0.341215	0.99999	-923.438	-79.3527	-9
Bm78	MaxEnv Pile_wave.WLC(1, 1),...	tauNxz [KPa]	MaxHotSpot	13	1	1	0.341215	0.99999	0.99999	79.3527	923.438	92
Bm78	MinEnv Pile_wave.WLC(1, 1),...	sigMxy [KPa]	MinHotSpot	19	7	19	0.99999	0.738187	0.99999	-12257.8	-1693.69	-1
Bm78	MaxEnv Pile_wave.WLC(1, 1),...	sigMxy [KPa]	MaxHotSpot	19	7	7	0.738187	0.99999	0.99999	1693.69	12257.8	12

Similarly, the tab "Beam Line Tables" lists all the detailed values in all hotspots at all positions.



Beam Line		Bm78				Bm78			
Load Case/Envelope		MinEnv Pile_wave.WLC(1, 1),...				MaxEnv Pile_wave.WLC(1, 1),...			
Component		tauNxz [KPa]				tauNxz [KPa]			
Hotspot		MinHotSpot				MaxHotSpot			
	Distance from End [m]		Hotspot	Load Case	Distance from End [m]		Hotspot	Load Case	Distance from End [m]
	0.000153247	-705.194		Pile_wave.WLC(3, 1)	0.000153247	705.194	3	Pile_wave.WLC(3, 1)	0.000153247
	0.668727	-664.884		Pile_wave.WLC(3, 1)	0.668727	664.884	3	Pile_wave.WLC(3, 1)	0.668727
	1.33745	-624.937		Pile_wave.WLC(3, 1)	1.33745	624.937	3	Pile_wave.WLC(3, 1)	1.33745
	2.00217	-591.397		Pile_wave.WLC(1, 1)	2.00217	591.397	3	Pile_wave.WLC(1, 1)	2.00217
	2.67491	-569.319		Pile_wave.WLC(1, 1)	2.67491	569.319	3	Pile_wave.WLC(1, 1)	2.67491

This page is blank for printing

5. REDESIGN

5.1 Introduction

The following list gives a brief overview of the steps involved in the design iteration process including member re-design (steps 7 – 9) and automatic reporting. The details are explained in the following chapters.

Steps in the re-iteration process	Comments
1. Make your model 2. Run analysis 3. Create capacity manager and make a code check run 4. Create members 5. Generate code check forces (code checking position) and execute the code check	**First pass** These steps are all part of the first pass. You always need to do this to find code check results and decide whether design iterations are needed.
6. Decide the content of a report (table of contents) and create (save) the report	The settings for the report are stored, and you can create a new report with the same table of content later.
7. Investigate and select members of interest. Right-click and select "Redesign" 8. Do your changes by altering the settings in the Redesign dialog. 9. Click "OK" when you want to use the settings you have changed	**These steps are also known as the re-design process.** Normally, you may need to loop some times over the steps 7 to 9 until you have an acceptable solution. Notice that the new code check results are not based on updated analysis results. Hence it is necessary to do a full re-run including a new code check to ensure consistent analysis and code check results.
10. Select "Run All"	This action will ensure consistency between capacity and structural models as well as do a full re-run of the analysis as well as activities pertaining to the code check execution. In case you have done a code check and modified a member (e.g. section or material data) outside the redesign feature, you can still click on "Run All" to easily get new code check results.
11. Check if the results are satisfactory, if not, go back to step 7.	
12. Auto-regenerate the report.	You can regenerate the report with the same table of content that you used in step 6.

The following limitations apply:

- If there are eccentricities to the beams, these need to be manually regenerated in case their section properties are modified during redesign. In such case you need to manually select "Update Structure from Members", apply the eccentricities before the Run All can be executed.
- In case you move or insert new members it is necessary to regenerate the capacity members.
- The redesign feature applies to members and not joints.

5.2 Redesign example

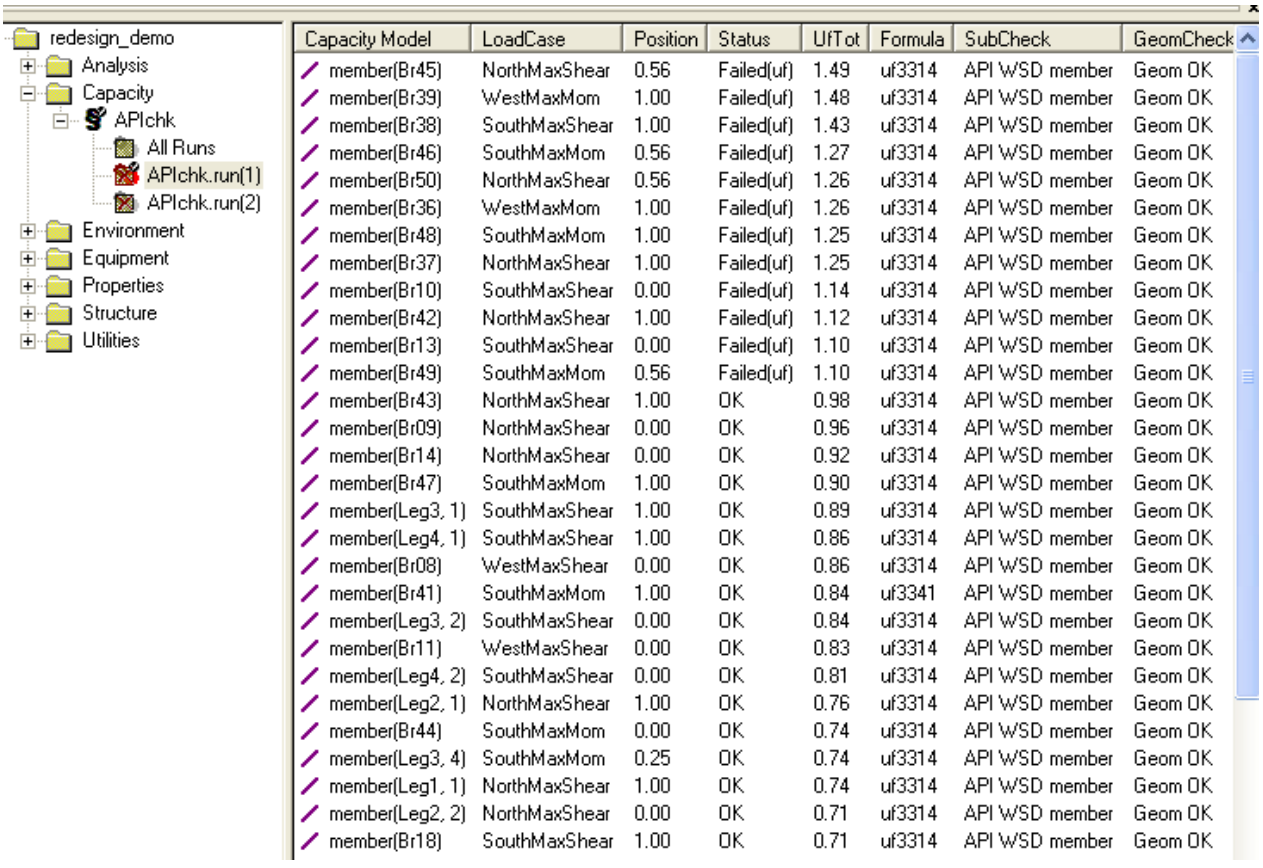
In order to present a *Redesign* example, a fully comprehensive example is carried out. From the Tutorial web page we select the “Jacket with tubular joints” model.

First we read in the following two .js files:

1. “GeniE_PSIanalysis_for_cc_in.js”
2. “GeniE_ccAPIWSD_in.js”

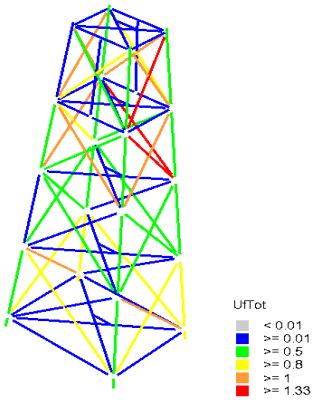
After loading the above mentioned .js files, we obtain a working model. The code check analysis has been run and we have two capacity managers.

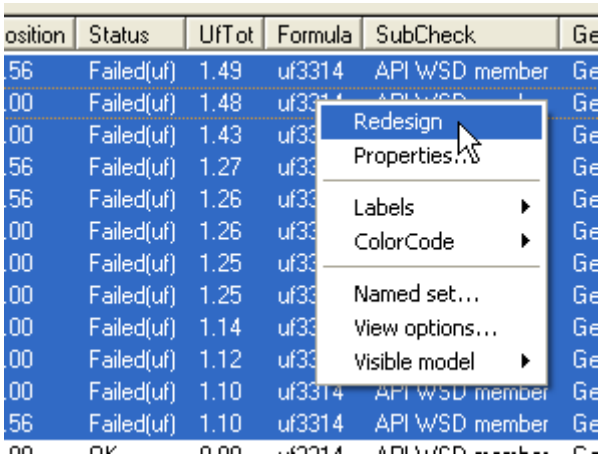
Taking a brief look at the results we can observe that for some members the usage factor (UfTot) is greater than 1, meaning that these structural members do not comply with the selected Specification Standard.



Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck
member(Br45)	NorthMaxShear	0.56	Failed(uf)	1.49	uf3314	API WSD member	Geom OK
member(Br39)	WestMaxMom	1.00	Failed(uf)	1.48	uf3314	API WSD member	Geom OK
member(Br38)	SouthMaxShear	1.00	Failed(uf)	1.43	uf3314	API WSD member	Geom OK
member(Br46)	SouthMaxMom	0.56	Failed(uf)	1.27	uf3314	API WSD member	Geom OK
member(Br50)	NorthMaxShear	0.56	Failed(uf)	1.26	uf3314	API WSD member	Geom OK
member(Br36)	WestMaxMom	1.00	Failed(uf)	1.26	uf3314	API WSD member	Geom OK
member(Br48)	SouthMaxMom	1.00	Failed(uf)	1.25	uf3314	API WSD member	Geom OK
member(Br37)	NorthMaxShear	1.00	Failed(uf)	1.25	uf3314	API WSD member	Geom OK
member(Br10)	SouthMaxShear	0.00	Failed(uf)	1.14	uf3314	API WSD member	Geom OK
member(Br42)	NorthMaxShear	1.00	Failed(uf)	1.12	uf3314	API WSD member	Geom OK
member(Br13)	SouthMaxShear	0.00	Failed(uf)	1.10	uf3314	API WSD member	Geom OK
member(Br49)	SouthMaxMom	0.56	Failed(uf)	1.10	uf3314	API WSD member	Geom OK
member(Br43)	NorthMaxShear	1.00	OK	0.98	uf3314	API WSD member	Geom OK
member(Br09)	NorthMaxShear	0.00	OK	0.96	uf3314	API WSD member	Geom OK
member(Br14)	NorthMaxShear	0.00	OK	0.92	uf3314	API WSD member	Geom OK
member(Br47)	SouthMaxMom	1.00	OK	0.90	uf3314	API WSD member	Geom OK
member(Leg3, 1)	SouthMaxShear	1.00	OK	0.89	uf3314	API WSD member	Geom OK
member(Leg4, 1)	SouthMaxShear	1.00	OK	0.86	uf3314	API WSD member	Geom OK
member(Br08)	WestMaxShear	0.00	OK	0.86	uf3314	API WSD member	Geom OK
member(Br41)	SouthMaxMom	1.00	OK	0.84	uf3341	API WSD member	Geom OK
member(Leg3, 2)	SouthMaxShear	0.00	OK	0.84	uf3314	API WSD member	Geom OK
member(Br11)	WestMaxShear	0.00	OK	0.83	uf3314	API WSD member	Geom OK
member(Leg4, 2)	SouthMaxShear	0.00	OK	0.81	uf3314	API WSD member	Geom OK
member(Leg2, 1)	NorthMaxShear	1.00	OK	0.76	uf3314	API WSD member	Geom OK
member(Br44)	SouthMaxMom	0.00	OK	0.74	uf3314	API WSD member	Geom OK
member(Leg3, 4)	SouthMaxMom	0.25	OK	0.74	uf3314	API WSD member	Geom OK
member(Leg1, 1)	NorthMaxShear	1.00	OK	0.74	uf3314	API WSD member	Geom OK
member(Leg2, 2)	NorthMaxShear	0.00	OK	0.71	uf3314	API WSD member	Geom OK
member(Br18)	SouthMaxShear	1.00	OK	0.71	uf3314	API WSD member	Geom OK

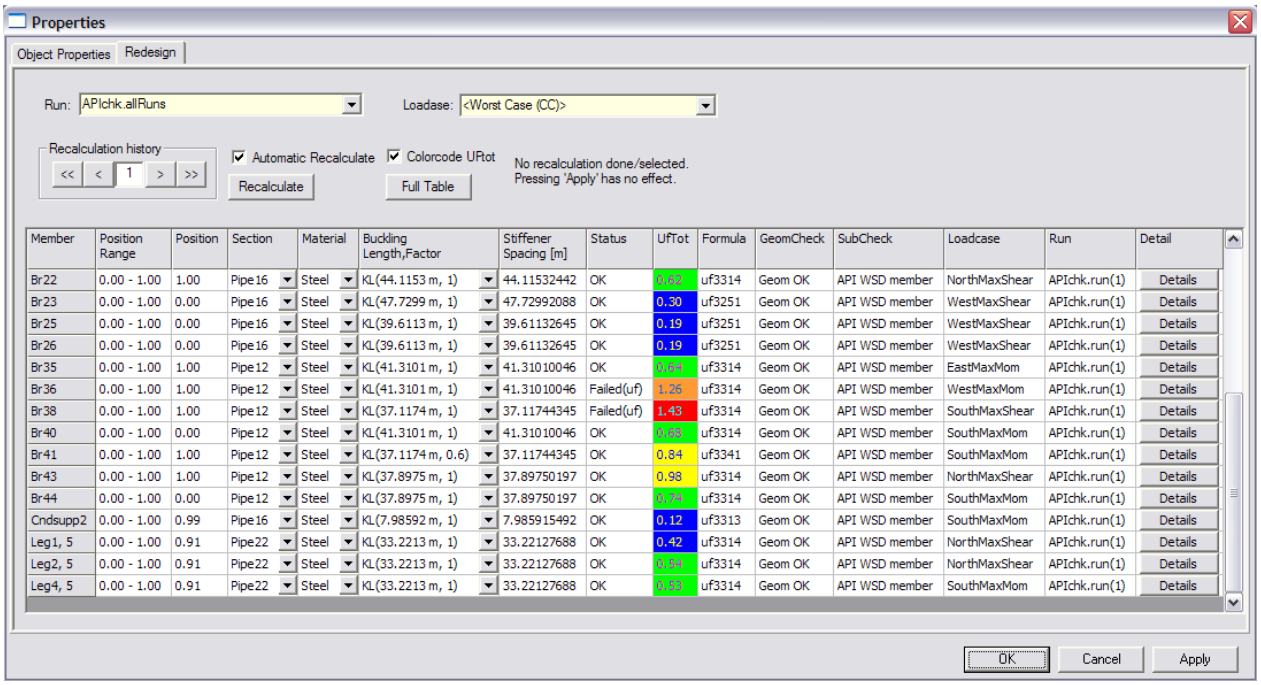
The usage factor for all the members from the run “APIchk.run(1)” can be colour coded as shown in the illustration to the right. We here show the model in the “Capacity Models” view. The same colours, representing the same range of UfTots, can be found in the Redesign window. This is a useful tool allowing the user to immediately spot the members that need to be updated.





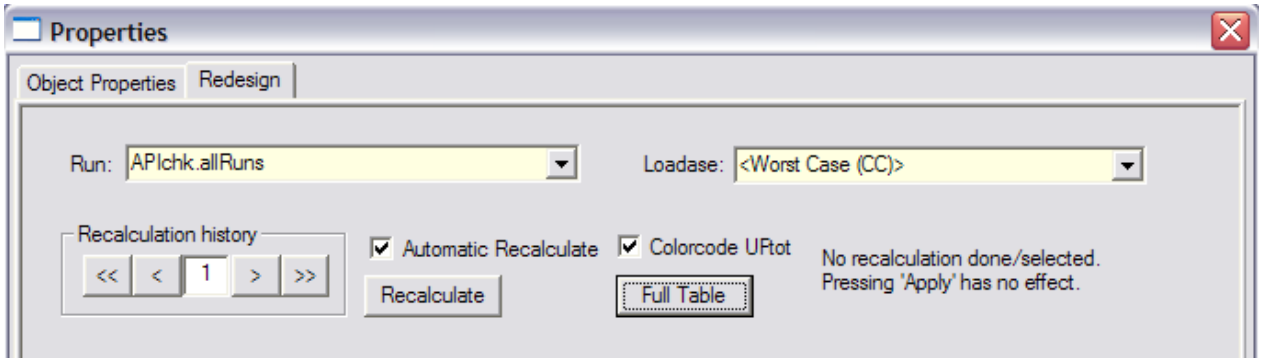
We select the members of interest. By rightclicking the member selection and selecting “Redesign” in the appearing menu, we open the Redesign dialog. The Redesign dialog contains the same members that we just selected..

The Redesign dialog appears:


Properties
Object Properties
Redesign
Run: APIchk.allRuns
Loadase: <Worst Case (CC)>
Recalculation history
Automatic Recalculate
Colorcode UFtot
No recalculation done/selected.
Pressing 'Apply' has no effect.
Recalculate
Full Table
Member | Position Range | Position | Section | Material | Buckling Length,Factor | Stiffener Spacing [m] | Status | UfTot | Formula | GeomCheck | SubCheck | Loadcase | Run | Detail
Br22 0.00 - 1.00 1.00 Pipe16 Steel KL(44.1153 m, 1) 44.11532442 OK 0.62 uf3314 Geom OK API WSD member NorthMaxShear APIchk.run(1) Details
Br23 0.00 - 1.00 0.00 Pipe16 Steel KL(47.7299 m, 1) 47.72992088 OK 0.30 uf3251 Geom OK API WSD member WestMaxShear APIchk.run(1) Details
Br25 0.00 - 1.00 0.00 Pipe16 Steel KL(39.6113 m, 1) 39.61132645 OK 0.19 uf3251 Geom OK API WSD member WestMaxShear APIchk.run(1) Details
Br26 0.00 - 1.00 0.00 Pipe16 Steel KL(39.6113 m, 1) 39.61132645 OK 0.19 uf3251 Geom OK API WSD member WestMaxShear APIchk.run(1) Details
Br35 0.00 - 1.00 1.00 Pipe12 Steel KL(41.3101 m, 1) 41.31010046 OK 0.83 uf3314 Geom OK API WSD member EastMaxMom APIchk.run(1) Details
Br36 0.00 - 1.00 1.00 Pipe12 Steel KL(41.3101 m, 1) 41.31010046 Failed(uf) 1.26 uf3314 Geom OK API WSD member WestMaxMom APIchk.run(1) Details
Br38 0.00 - 1.00 1.00 Pipe12 Steel KL(37.1174 m, 1) 37.11744345 Failed(uf) 1.43 uf3314 Geom OK API WSD member SouthMaxShear APIchk.run(1) Details
Br40 0.00 - 1.00 0.00 Pipe12 Steel KL(41.3101 m, 1) 41.31010046 OK 0.62 uf3314 Geom OK API WSD member SouthMaxMom APIchk.run(1) Details
Br41 0.00 - 1.00 1.00 Pipe12 Steel KL(37.1174 m, 0.6) 37.11744345 OK 0.84 uf3341 Geom OK API WSD member SouthMaxMom APIchk.run(1) Details
Br43 0.00 - 1.00 1.00 Pipe12 Steel KL(37.8975 m, 1) 37.89750197 OK 0.98 uf3314 Geom OK API WSD member NorthMaxShear APIchk.run(1) Details
Br44 0.00 - 1.00 0.00 Pipe12 Steel KL(37.8975 m, 1) 37.89750197 OK 0.76 uf3314 Geom OK API WSD member SouthMaxMom APIchk.run(1) Details
Cndsupp2 0.00 - 1.00 0.99 Pipe16 Steel KL(7.98592 m, 1) 7.985915492 OK 0.12 uf3313 Geom OK API WSD member SouthMaxMom APIchk.run(1) Details
Leg1, 5 0.00 - 1.00 0.91 Pipe22 Steel KL(33.2213 m, 1) 33.22127688 OK 0.42 uf3314 Geom OK API WSD member NorthMaxShear APIchk.run(1) Details
Leg2, 5 0.00 - 1.00 0.91 Pipe22 Steel KL(33.2213 m, 1) 33.22127688 OK 0.83 uf3314 Geom OK API WSD member NorthMaxShear APIchk.run(1) Details
Leg4, 5 0.00 - 1.00 0.91 Pipe22 Steel KL(33.2213 m, 1) 33.22127688 OK 0.83 uf3314 Geom OK API WSD member SouthMaxMom APIchk.run(1) Details
OK
Cancel
Apply

In order to explain the features of this *Redesign* dialog a split between the header of the dialog and the grid is made.

The header is presented in the figure below.



The *Run* combo box allows you to select which run (single run or all runs at once) you want to include in the redesign. Note that if you don't specify a run, *all* the runs will be taken into consideration for the redesign process and the highest usage factor among all the runs will be shown.

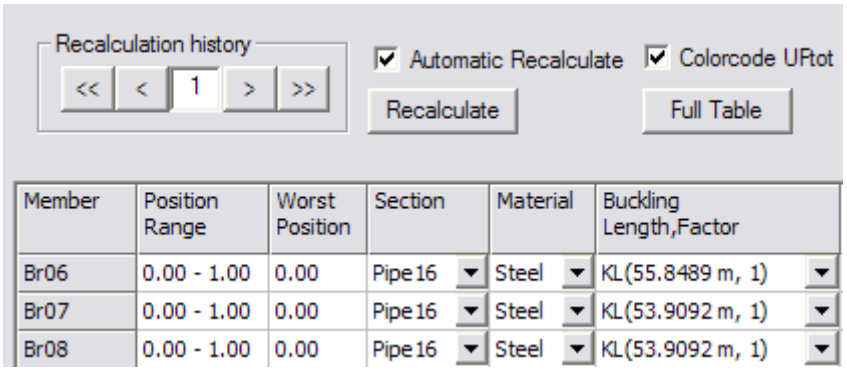
In the *Loadcase* combo box you can select one specific loadcase, or you can select *Worst* Case, giving you the worst results for all the loadcases.

The *Recalculation history* tracks the changes you do in each step of your redesign process.

The first button, on the left hand side of the counter, presents the initial design properties.

The second button presents the previous design iteration properties.

In the same way, the second button on the right hand side shows the last design iteration and the first button the next design iteration counting from the design iteration displayed on the recalculation history counter.



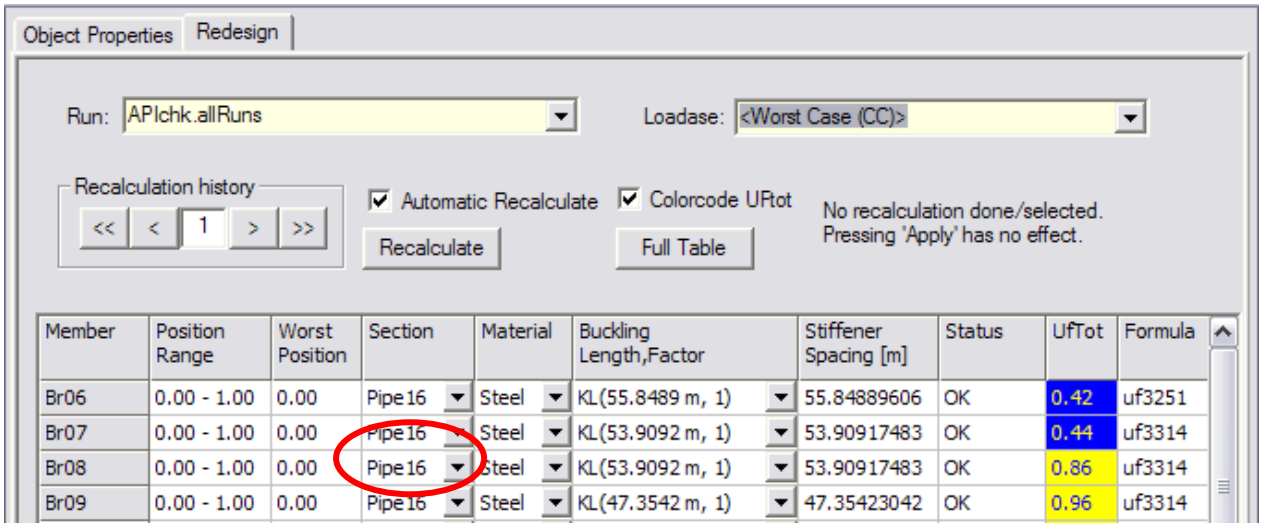
Note that a new step is added to the *Recalculation history* whenever you make a change. You can move backwards and forwards through the Recalculation History if the results of your changes are not satisfactory, or if you just want to try different settings.

The *Automatic Recalculate* check box and the *Recalculate* button are parts of the same purpose: to set a new iteration design using the new design parameters. If the check box is checked the *Recalculate* is overridden and the new usage factors and displayed properties are automatically updated.

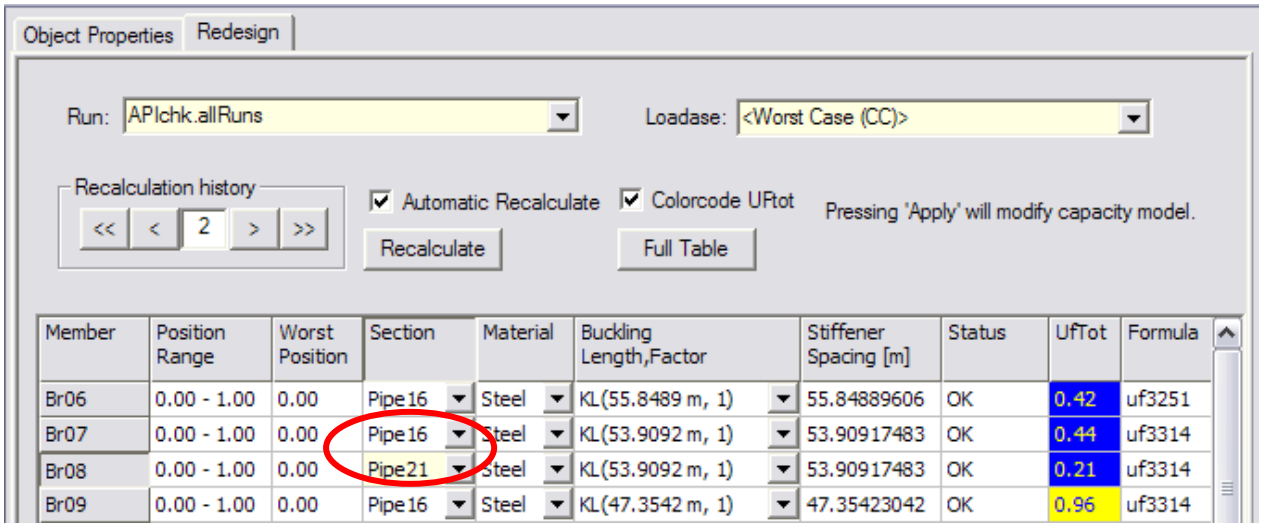
If the *Automatic Recalculate* check box is not selected, you must click the *Recalculate* button every time a new iteration design is necessary.

Also note that the background colour changes from white to light yellow when a design parameter is changed. This is shown in the illustrations below.

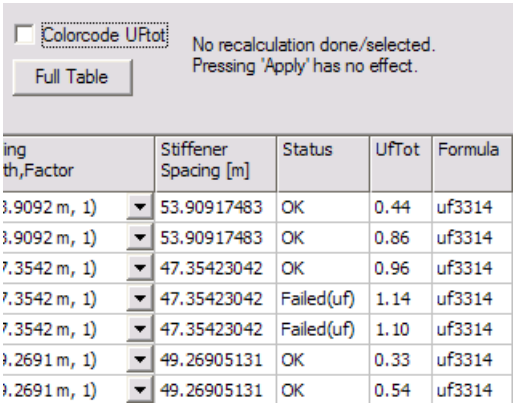
Before changing the Section property of Br08:


Object Properties
Redesign
Run: APIchk.allRuns
Loadase: <Worst Case (CC)>
Recalculation history
<< < 1 > >>
Automatic Recalculate
Colorcode UFtot
Recalculate
Full Table
No recalculation done/selected.
Pressing 'Apply' has no effect.
Member | Position Range | Worst Position | Section | Material | Buckling Length,Factor | Stiffener Spacing [m] | Status | UfTot | Formula
Br06 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(55.8489 m, 1) | 55.84889606 | OK | 0.42 | uf3251
Br07 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(53.9092 m, 1) | 53.90917483 | OK | 0.44 | uf3314
Br08 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(53.9092 m, 1) | 53.90917483 | OK | 0.86 | uf3314
Br09 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(47.3542 m, 1) | 47.35423042 | OK | 0.96 | uf3314

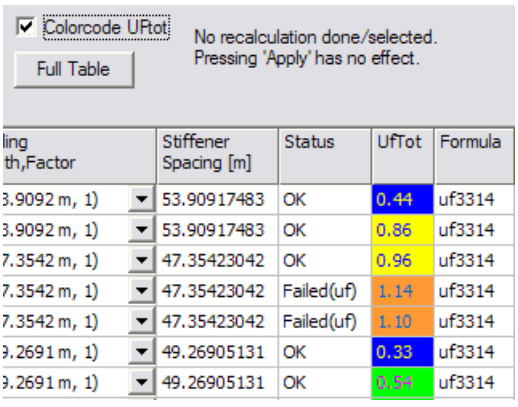
After changing the Section property of Br08 from Pipe16 to Pipe21, the background colour has changed from white to light yellow.


Object Properties
Redesign
Run: APIchk.allRuns
Loadase: <Worst Case (CC)>
Recalculation history
<< < 2 > >>
Automatic Recalculate
Colorcode UFtot
Recalculate
Full Table
Pressing 'Apply' will modify capacity model.
Member | Position Range | Worst Position | Section | Material | Buckling Length,Factor | Stiffener Spacing [m] | Status | UfTot | Formula
Br06 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(55.8489 m, 1) | 55.84889606 | OK | 0.42 | uf3251
Br07 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(53.9092 m, 1) | 53.90917483 | OK | 0.44 | uf3314
Br08 | 0.00 - 1.00 | 0.00 | Pipe21 | Steel | KL(53.9092 m, 1) | 53.90917483 | OK | 0.21 | uf3314
Br09 | 0.00 - 1.00 | 0.00 | Pipe16 | Steel | KL(47.3542 m, 1) | 47.35423042 | OK | 0.96 | uf3314

The Colorcode UFtot check box allows you to display *UfTot* with the same colour code as the colour code used on the capacity models result. This is useful in order to inspect which structural elements are prone to fail.



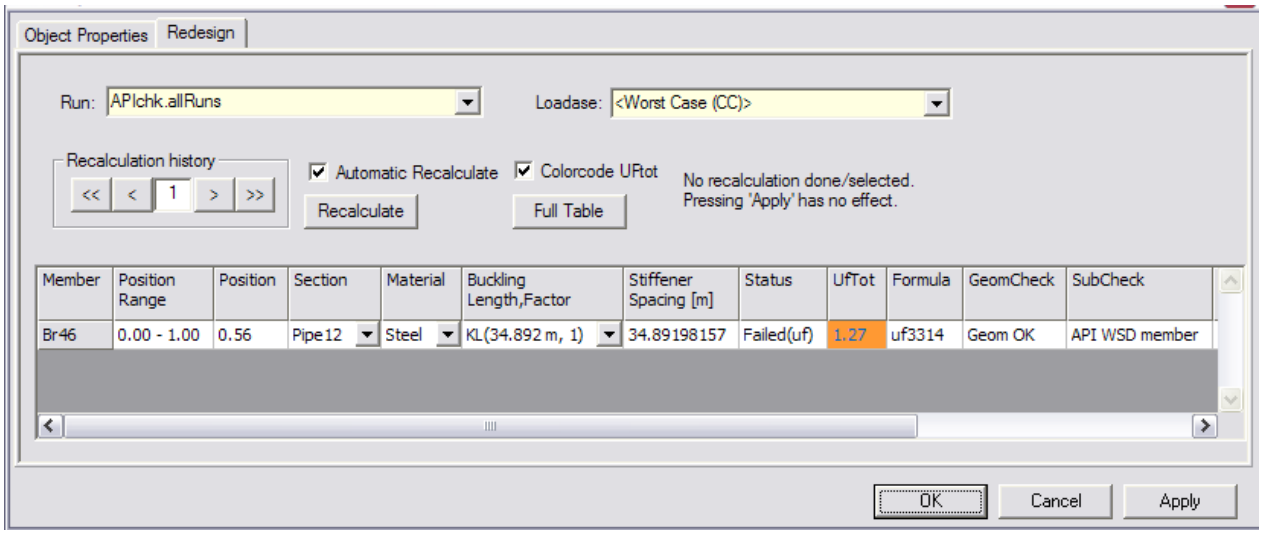
ing th,Factor	Stiffener Spacing [m]	Status	UfTot	Formula
3.9092 m, 1)	53.90917483	OK	0.44	uf3314
3.9092 m, 1)	53.90917483	OK	0.86	uf3314
7.3542 m, 1)	47.35423042	OK	0.96	uf3314
7.3542 m, 1)	47.35423042	Failed(uf)	1.14	uf3314
7.3542 m, 1)	47.35423042	Failed(uf)	1.10	uf3314
9.2691 m, 1)	49.26905131	OK	0.33	uf3314
9.2691 m, 1)	49.26905131	OK	0.54	uf3314



ling th,Factor	Stiffener Spacing [m]	Status	UfTot	Formula
3.9092 m, 1)	53.90917483	OK	0.44	uf3314
3.9092 m, 1)	53.90917483	OK	0.86	uf3314
7.3542 m, 1)	47.35423042	OK	0.96	uf3314
7.3542 m, 1)	47.35423042	Failed(uf)	1.14	uf3314
7.3542 m, 1)	47.35423042	Failed(uf)	1.10	uf3314
9.2691 m, 1)	49.26905131	OK	0.33	uf3314
9.2691 m, 1)	49.26905131	OK	0.54	uf3314

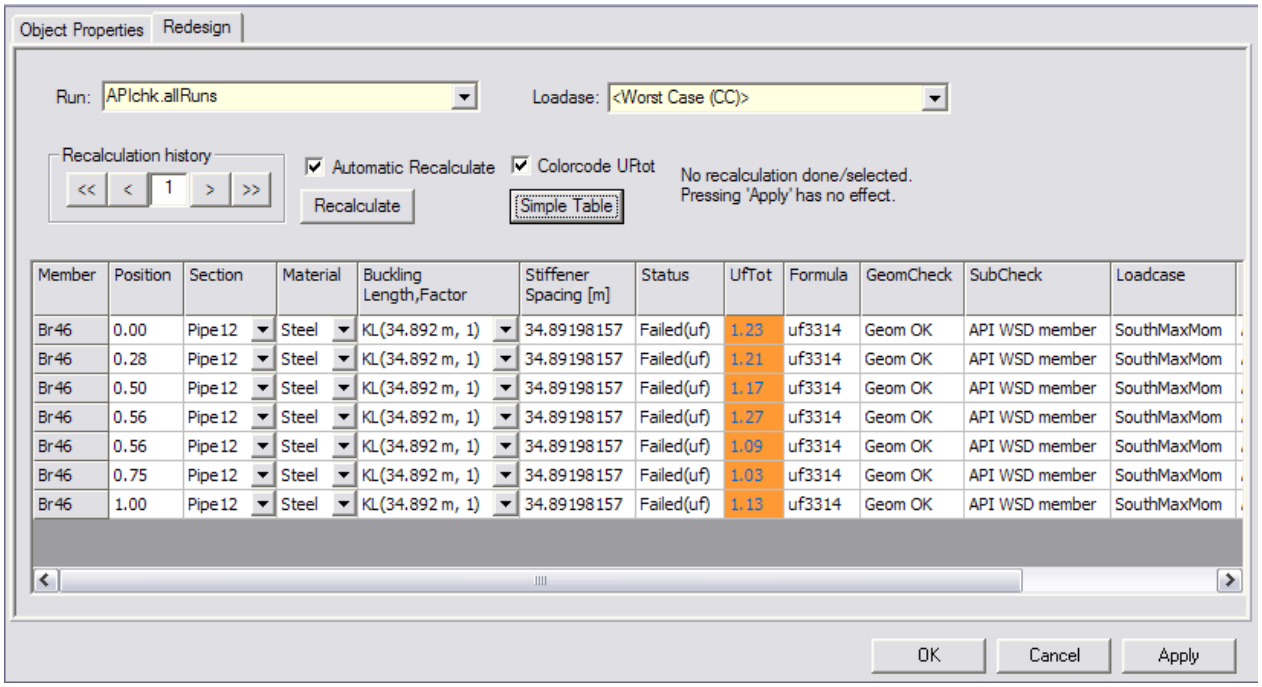
By pressing the *Full Table* button, you can get more detailed information about your members. This allows you to access information about the structural capacity model from all the checked points within the capacity model.

The following two illustrations demonstrate the use of this button for a single capacity member. First we look at the member "Br46" in the Simple Table view. The member is displayed on one single line showing the result for the worst position:



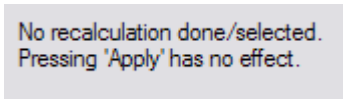
Member	Position Range	Position	Section	Material	Buckling Length,Factor	Stiffener Spacing [m]	Status	UfTot	Formula	GeomCheck	SubCheck
Br46	0.00 - 1.00	0.56	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.27	uf3314	Geom OK	API WSD member

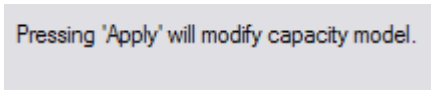
After clicking the *Full Table* button, the results for the single member, "Br46", is displayed for every checked position along the member as can be seen in the illustration below.



Member	Position	Section	Material	Buckling Length,Factor	Stiffener Spacing [m]	Status	UfTot	Formula	GeomCheck	SubCheck	Loadcase
Br46	0.00	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.23	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	0.28	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.21	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	0.50	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.17	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	0.56	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.27	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	0.56	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.09	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	0.75	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.03	uf3314	Geom OK	API WSD member	SouthMaxMom
Br46	1.00	Pipe12	Steel	KL(34.892 m, 1)	34.89198157	Failed(uf)	1.13	uf3314	Geom OK	API WSD member	SouthMaxMom

Information Window: This area shows useful information about the *Redesign* process and updates to the capacity model.

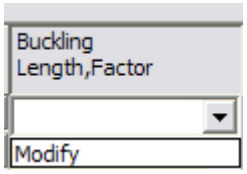




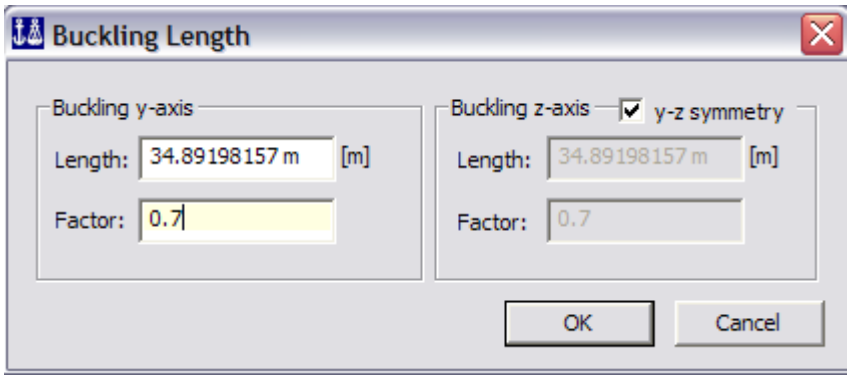
Since the ends of the members are fixed, we want to reduce the buckling factor from 1 to 0.7, just as an example. This change can be done on both buckling length properties: buckling length or buckling factor. Next illustrations show the update done to the buckling factor.

Buckling Length,Factor	Stiffener Spacing [m]	Status	UfTot	Formula
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.23	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.21	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.17	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.27	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.09	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.03	uf3314
KL(34.892 m, 1)	34.89198157	Failed(uf)	1.13	uf3314

After clicking *Buckling Length, Factor* combo box, select *Modify*,



and the following dialog box appears.

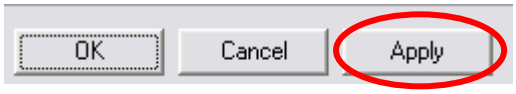


After updating the fields click *OK* and the changes will be effective to the code check.

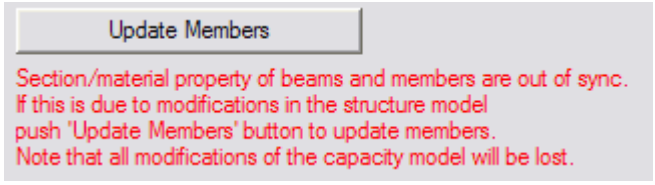
The UfTot for all the members are now below 1.00.

Buckling Length,Factor	Stiffener Spacing [m]	Status	UfTot	Formula
KL(34.892 m, 0.7)	34.89198157	OK	0.89	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.88	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.88	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.89	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.83	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.82	uf3314
KL(34.892 m, 0.7)	34.89198157	OK	0.84	uf3314

If the changes operated on the code check analysis are according with your aim, you should press *Apply* in order to update the capacity model. *OK* also accepts your changes to the capacity model.



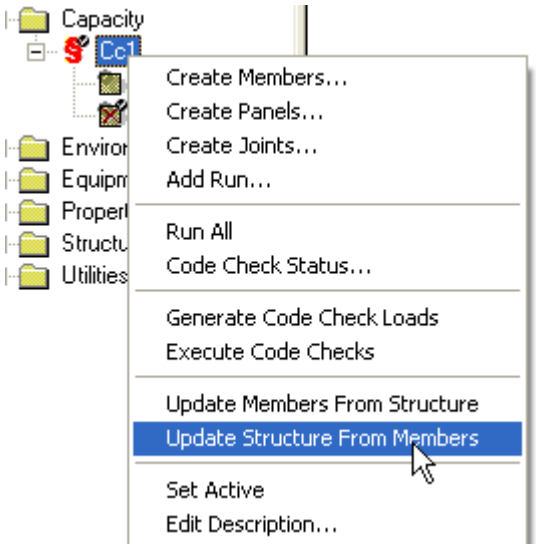
If you intend to continue the Redesign process for other structural members, the following message pops up at the *Information Window*.


Update Members
Section/material property of beams and members are out of sync.
If this is due to modifications in the structure model
push 'Update Members' button to update members.
Note that all modifications of the capacity model will be lost.

In order to update the structure concept model, commit the changes performed during redesign by clicking "Update Structure From Members"

"Update Members from Structure" is used when you change something in the concept model, like removing a beam or altering segmentation, and you want this change to be reflected in the capacity model. Information is moved ***from*** the concept model ***to*** the capacity model.

"Update Structure From Member" is used to move information ***from*** the capacity model ***to*** the concept model. This is further explained below:

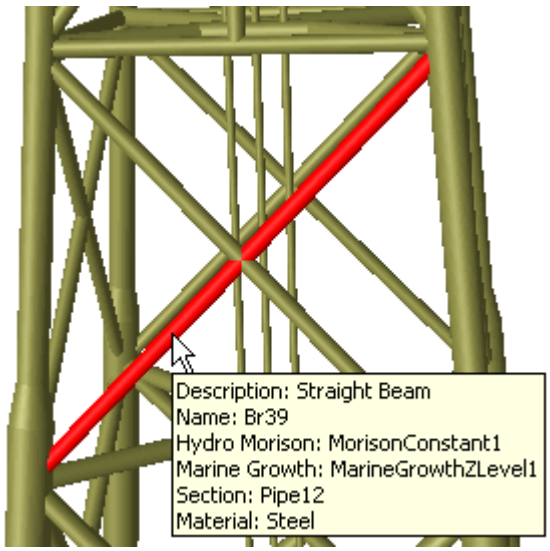


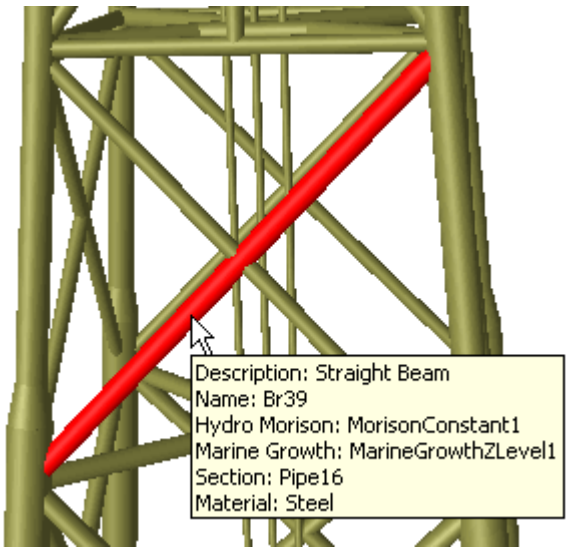
Earlier in this example, we replaced the section of the member Br39. It was originally Pipe12, and we replaced it by Pipe16. If we look at Br39 in the graphical view (Modeling – Structure) we see that it still has its original section, Pipe12.

To change it to its new section we need to rightclick the capacity manager and select "Update Structure From Member".

Below you can see how the section of member Br39 is replaced. The figures show the beam before and after clicking "Update Structure From Member".

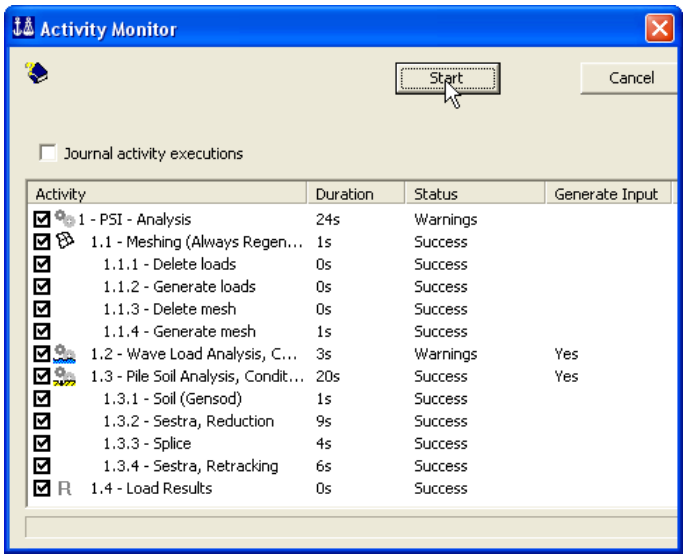
The description shows that the section has been replaced, and the replacement is also visible in the graphical view as the beam looks thicker.

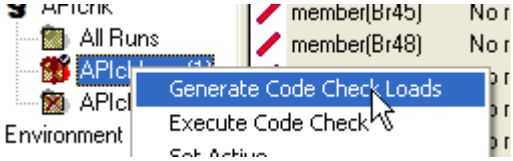




Be careful not to mix up the terms “Update Members From Structure” and “Update Structure From Members”.

Finally, we want to re-run the analysis and update the capacity manager to see the correct UfTot for the members. Click “Start” on the Activity Monitor for the analysis and select “Generate Code Check Loads” and “Execute Code Check” for the capacity manager.


Activity Monitor
Start
Cancel
Journal activity executions
Activity
Duration
Status
Generate Input
1 - PSI - Analysis 24s Warnings
1.1 - Meshing (Always Regen... 1s Success
1.1.1 - Delete loads 0s Success
1.1.2 - Generate loads 0s Success
1.1.3 - Delete mesh 0s Success
1.1.4 - Generate mesh 1s Success
1.2 - Wave Load Analysis, C... 3s Warnings Yes
1.3 - Pile Soil Analysis, Condit... 20s Success Yes
1.3.1 - Soil (Gensod) 1s Success
1.3.2 - Sestra, Reduction 9s Success
1.3.3 - Splice 4s Success
1.3.4 - Sestra, Retracking 6s Success
1.4 - Load Results 0s Success

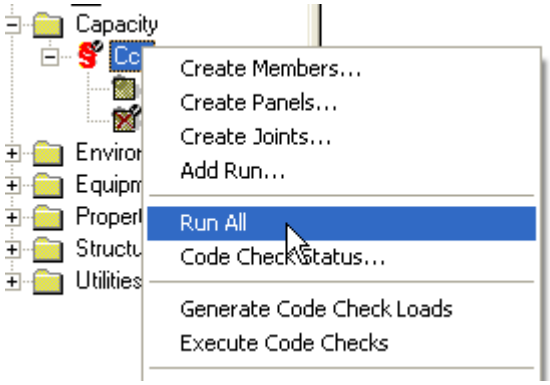


The UfTots are almost the same as before running the analysis again, but there are some differences. Observe that for instance UfTot for member Br43 has changed from 0.98 to 0.97.

Capacity Model	LoadCase	Position	Status	UfTot	Formula	Su
member(Br48)	SouthMaxMom	1.00	OK	0.99	uf3314	AF
member(Br43)	NorthMaxShear	1.00	OK	0.97	uf3314	AF
member(Br09)	NorthMaxShear	0.00	OK	0.96	uf3314	AF
member(Br38)	SouthMaxMom	0.00	OK	0.93	uf3314	AF
member(Br14)	NorthMaxShear	0.00	OK	0.93	uf3314	AF
member(Br45)	NorthMaxShear	0.56	OK	0.92	uf3314	AF
member(Leg3, 1)	SouthMaxShear	1.00	OK	0.89	uf3314	AF
member(Br46)	SouthMaxMom	0.00	OK	0.89	uf3314	AF
member(Br47)	SouthMaxMom	1.00	OK	0.88	uf3314	AF

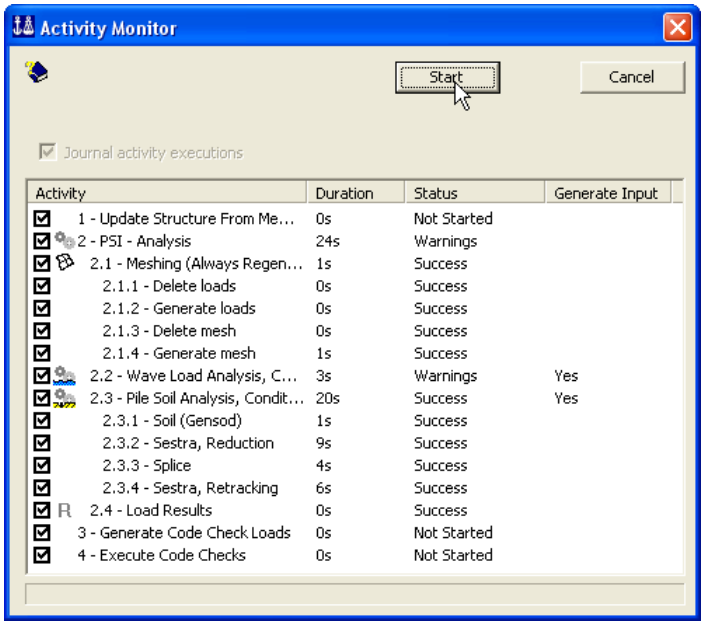
You may want to re-run the analysis, generate the code check loads and execute the code check in one single operation.

This is possible if you rightclick the capacity manager and select “Run All”.

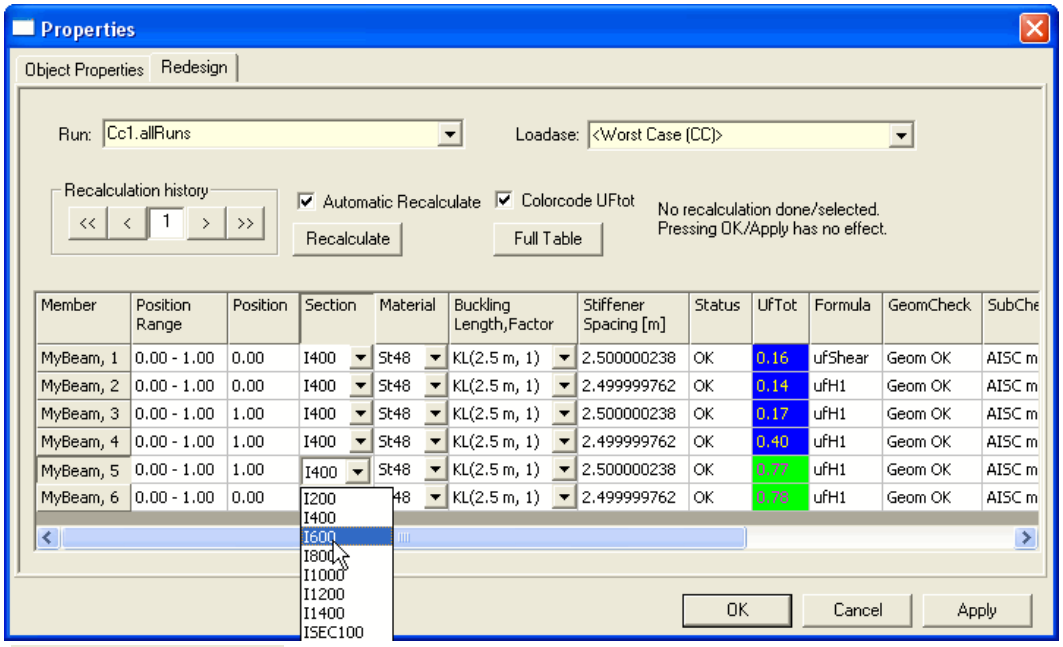


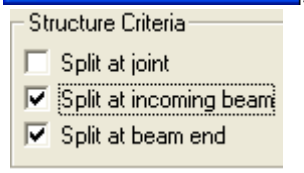
The activity monitor appears. Note that step 3, “Generate Code Check Loads” and step 4, “Execute Code Check Loads” are now included.

Just click Start to see the results of your changes.

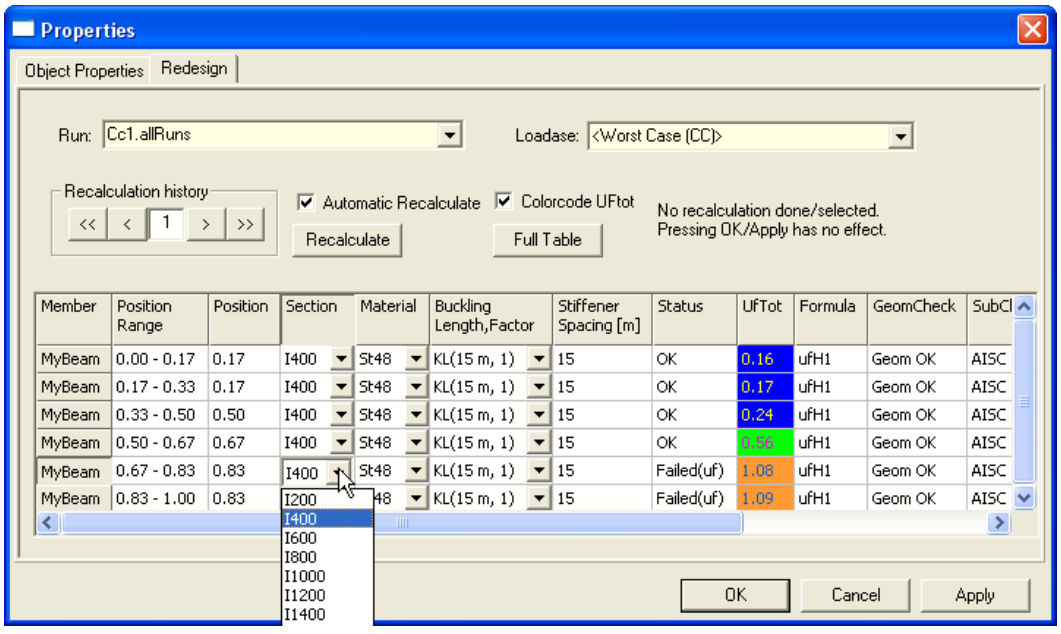

Activity Monitor
Start
Cancel
Journal activity executions
Activity
Duration
Status
Generate Input
1 - Update Structure From Me... 0s Not Started
2 - PSI - Analysis 24s Warnings
2.1 - Meshing (Always Regen... 1s Success
2.1.1 - Delete loads 0s Success
2.1.2 - Generate loads 0s Success
2.1.3 - Delete mesh 0s Success
2.1.4 - Generate mesh 1s Success
2.2 - Wave Load Analysis, C... 3s Warnings Yes
2.3 - Pile Soil Analysis, Condit... 20s Success Yes
2.3.1 - Soil (Gensod) 1s Success
2.3.2 - Sestra, Reduction 9s Success
2.3.3 - Splice 4s Success
2.3.4 - Sestra, Retracking 6s Success
2.4 - Load Results 0s Success
3 - Generate Code Check Loads 0s Not Started
4 - Execute Code Checks 0s Not Started

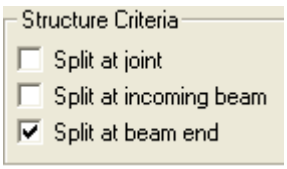
The beams in the example consisted of only one segment. It is also possible do redesign on segmented beams. This is shown in the illustrations below.


Properties
Object Properties
Redesign
Run: Cc1.allRuns
Loadase: <Worst Case (CC)>
Recalculation history
Automatic Recalculate
Colorcode UFtot
Recalculate
Full Table
No recalculation done/selected.
Pressing OK/Apply has no effect.
Member
Position Range
Position
Section
Material
Buckling Length,Factor
Stiffener Spacing [m]
Status
UfTot
Formula
GeomCheck
MyBeam, 1 0.00 - 1.00 0.00 I400 St48 KL(2.5 m, 1) 2.500000238 OK 0.16 ufShear Geom OK AISC m
MyBeam, 2 0.00 - 1.00 0.00 I400 St48 KL(2.5 m, 1) 2.499999762 OK 0.14 ufH1 Geom OK AISC m
MyBeam, 3 0.00 - 1.00 1.00 I400 St48 KL(2.5 m, 1) 2.500000238 OK 0.17 ufH1 Geom OK AISC m
MyBeam, 4 0.00 - 1.00 1.00 I400 St48 KL(2.5 m, 1) 2.499999762 OK 0.40 ufH1 Geom OK AISC m
MyBeam, 5 0.00 - 1.00 1.00 I400 St48 KL(2.5 m, 1) 2.500000238 OK 0.77 ufH1 Geom OK AISC m
MyBeam, 6 0.00 - 1.00 0.00 48 KL(2.5 m, 1) 2.499999762 OK 0.78 ufH1 Geom OK AISC m
I200
I400
I600
I800
I1000
I1200
I1400
ISEC100
OK
Cancel
Apply



The beam "MyBeam" is made up of six segments. You can change the properties of each segment individually. The illustration above shows how the redesign dialog will appear when "Split at incoming beam" is checked in the "Create members" dialog. Compare it with the illustration below.





The illustration above shows how the redesign dialog will appear when "Split at incoming beam" is left unchecked in the "Create members" dialog. This is the same "MyBeam" as in the first illustration.

It is important to note that "Buckling Length" and "Stiffener Spacing" differ from the first illustration. This also affects the UfTot.

6. POST PROCESSING OF SUPERELEMENT RESULTS

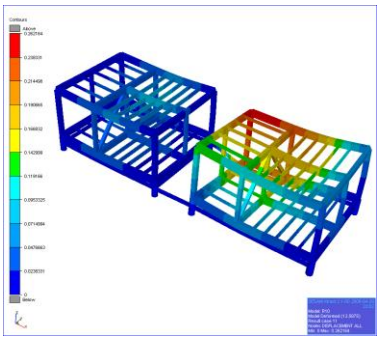
From GeniE version 5.3, the handling of imported SIN results from an external analysis has been modified. Result cases are now separate entities.

GeniE may be used to create first level elements (so-called superelements) that can be used in a superelement analysis. In such case the entire process should be modelled and executed from Sesam Brix Explorer. Depending on the tasks to be performed other Sesam modules, in addition to GeniE, should also be used to assemble and analyse the structure.

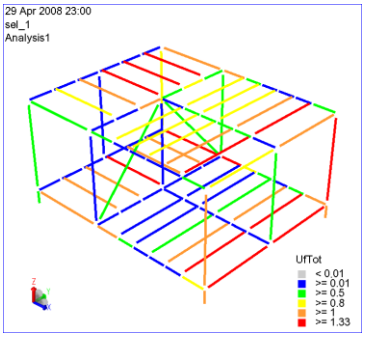
It is also possible to import results from a superelement analysis back to a first level element created by GeniE to do post-processing like code checking. The procedure and requirements to do so are explained in the following using a practical example.

The basic requirements are:

- Import of results can be performed on models created and meshed in GeniE. It is not possible to use repetitions of superelements or merge result files when importing results.
- Post-processing (code checking, stress/force/displacement evaluations) can be done based on a static or time domain analysis.
- When you have exported a finite element model (***File|Export|FEM File***) you cannot modify the original model (if you do, the new mesh must be re-exported). When importing the results, GeniE will check that the number of nodes and finite elements are the same for the original FEM file and the imported result file. Furthermore, the date identification card will be checked to see if there have been any alterations. This is a safety check to ensure that the results refer to the same FEM model created by GeniE.
- In a single superelement, static analysis, there will be a 1:1 relation between finite element loadcase numbers and finite element result-case numbers. Observe that this refers to the numeric reference system used on the FEM/SIN files, and should not be confused with the numbers used in the loadcase names in GeniE.
- Load combinations can be defined in the superelement assembly. When the proper *RSEL* option is used in SESTRA, this will ensure that bending moments and stresses for beams will also correctly take applied beam loads into account (automatically handled if analysis is performed using defaults in Brix Explorer configured for Sesam)
- Result combinations can be made in Prepost after analysis in GeniE based on analysis results. Note that GeniE version 5.3 does not support creation of combinations of result cases inside GeniE.
- It is also possible to do load combinations using a more traditional approach where the load combinations are done prior to analysis. Both alternatives are shown in the following.



Displacements for global model from a superelement analysis. Shown in Xtract

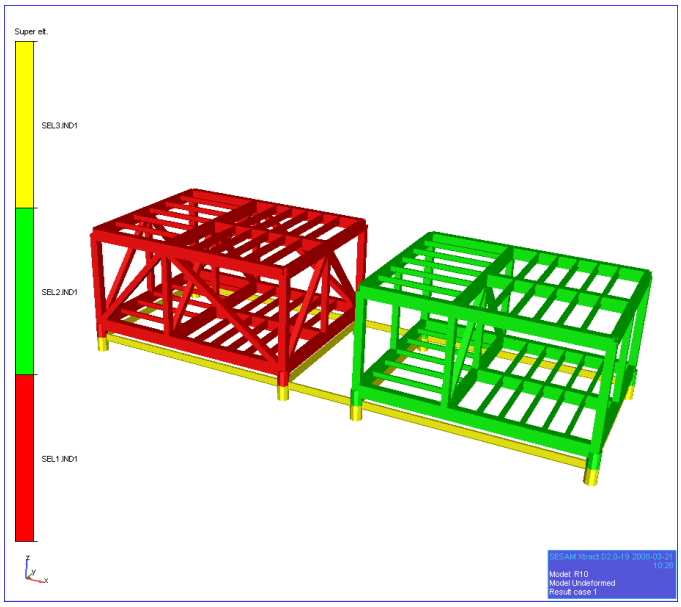


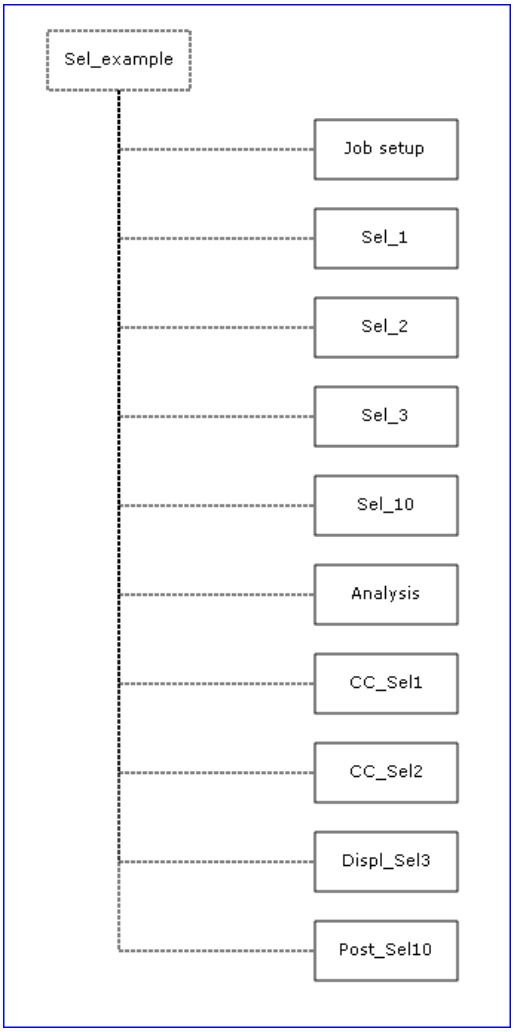
Code checking performed on a superelement part of a superelement. Shown in GeniE

6.1 Modelling and post-processing

The following example is used to show how to build a small superelement analysis and how to code check individual parts of it using Brix Explorer configured for Sesam as a process modelling tool. For details on how to use Brix Explorer, please consult its user manual.

The structure consists of 3 individual parts (Sel_1, Sel_2 and Sel_3). Presel is used to assemble the individual parts into Sel_10 and Sestra is used to analyse it. Code checking and post-processing of individual parts is done using GeniE (CC_Sel1, CC_Sel2 and Displ_Sel3). Finally the entire structure can be post-processed using Xtract (Post_Sel10).





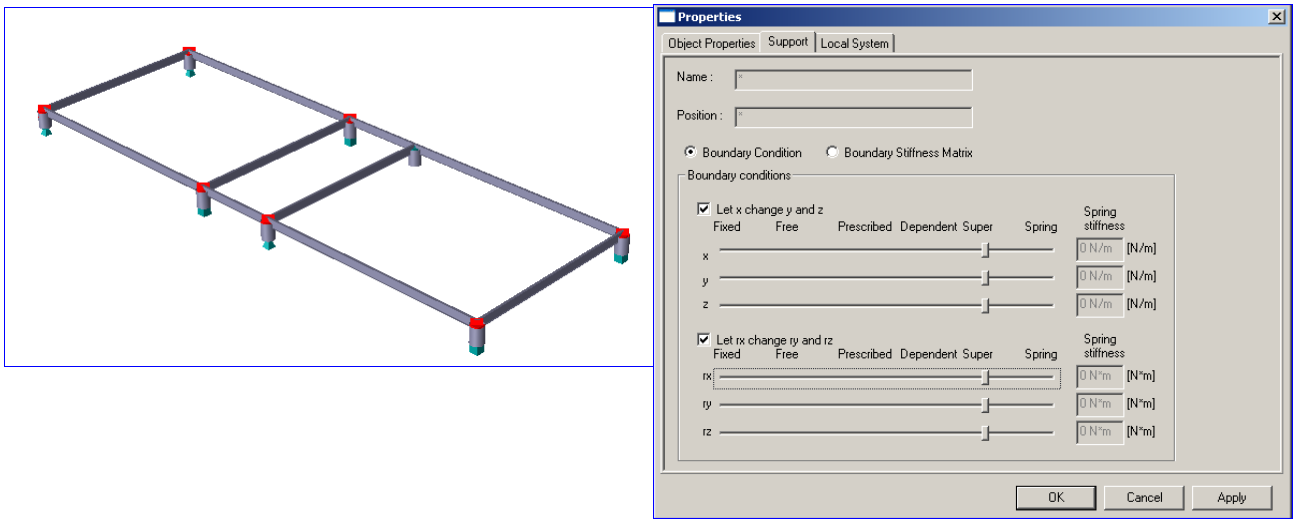
There are 4 global load cases in the superelement analysis. These are built by referencing basic load cases in each of the first level elements defined by GeniE.

For details on how to assemble structural parts as well as creating load combinations, reference is made to Presel User Manual.

6.1.1 GeniE first level superelements

When making a superelement you need to specify super-nodes to connect to other structural parts. Boundary conditions to prevent rigid body motions may be defined in a first level superelement or at higher order elements.

The highlighted support points below for Sel_3 are specified with super degree of freedom to connect the frames Sel_1 and Sel_2. The relevant positions for Sel_1 and Sel_2 are also specified with super degree of freedom.

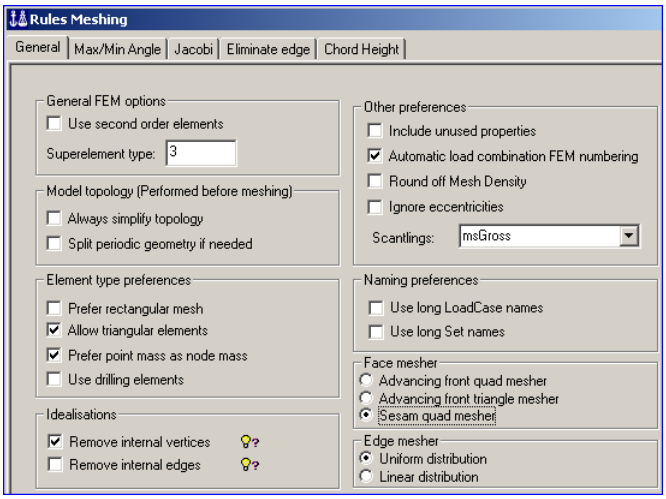


Each of the superelements has 11 loadcases, for further details see the following section on global load combination.

The finite element model must be defined from an analysis activity – this is required to set up the necessary folders for results handling after the analysis. To make the mesh and export it the following sequence can be used:

6.1.1.1 Define the superelement number

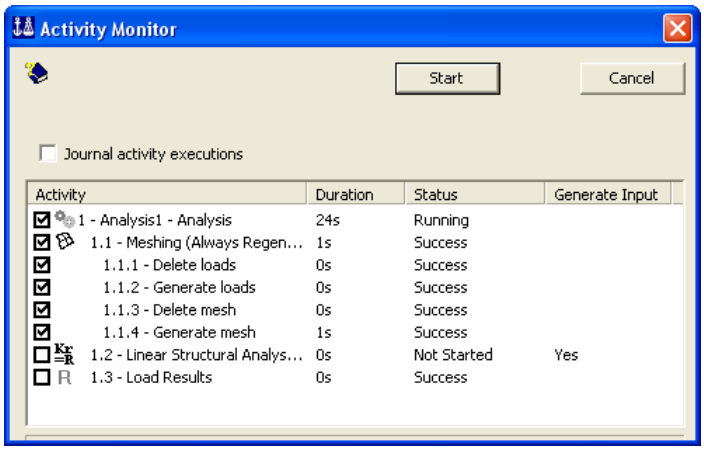
The superelement number is specified from Edit|Rules|Meshing. In this case the number is 3.



6.1.1.2 Make the finite element model

The finite element mesh is created from the Analysis browser or by using ***ALT+D***. In the example to the right meshing is the only activity to be performed.

It is required that the finite element model is created from the activity monitor as this will create an analysis folder for the model – this folder is used when loading the results back to GeniE from the superelement analysis.



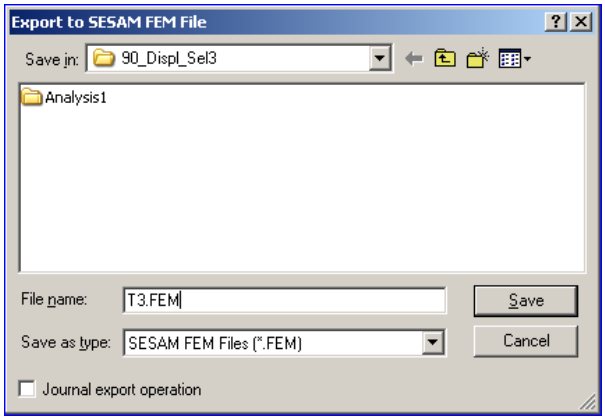
Superelement load combinations are not defined in GeniE, they are defined in Presel. The combinations on the highest superelement level will define loadcases that are retracked, also referred to as result-cases.

There is also an alternative method:

Use result combinations in Prepost. In this scenario, each of the 1st level loadcases are combined in Presel with a factor of 1.0 up to the top level.

6.1.1.3 Export the finite element model

The finite element model is exported from ***File|Export|FEM File***. In this case the superelement model is termed T3.FEM.



6.1.1.4 Make the global load combinations – combine results in Prepost

The following tables show how it is possible to model only the relevant basic loadcases in each of the GeniE models, and then combine these in Presel, maintaining the individual loads separately. 3 superelements with 3, 3 and 1 loadcases respectively, end up as 7 top level loadcases. These 7 loadcases are solved in Sestra.

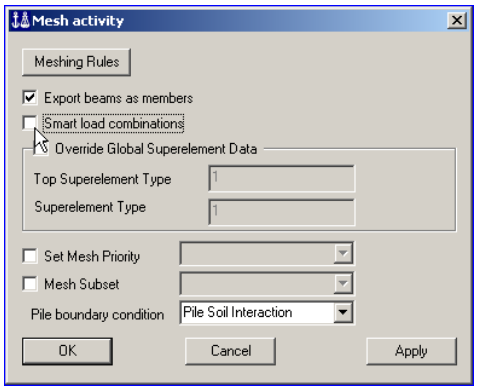
First lift each basic loadcase up to the highest superelement level by means of Presel load combinations, using a load factor of 1.0.

S.el. 10	S.el. 1		S.el. 2		S.el. 3	
Presel GLC	BL (LLC)	Content	BL (LLC)	Content	BL (LLC)	Content
1	BL1 (1)	Self-weight				
2	BL2 (2)	Equipments				
3	BL3 (3)	Line loads				
4			BL1 (4)	Self-weight		
5			BL2 (5)	Equipments		
6			BL3 (6)	Line loads		
7					BL1 (7)	Self-weight

Then, after running Sestra, creating result combinations in Prepost to achieve the required combinations needed for post processing, and importing the results - ***File\|Import\|External Results SIN file.*** Post-processing can now be done e.g. as shown in Section 5.1.2.

GeniE	Presel GLC	Comment
=====	=====	Original S.el. 1 load cases follow
resultCase(1)	1	results from BL1 in Sel1
resultCase(2)	2	results from BL2 in Sel1
resultCase(3)	3	results from BL3 in Sel1
=====	=====	Original S.el. 2 load cases follow
resultCase(4)	4	results from BL1 in Sel2
resultCase(5)	5	results from BL2 in Sel2
resultCase(6)	6	results from BL3 in Sel2
=====	=====	Original S.el. 3 load cases follow
resultCase(7)	7	results from BL1 in Sel3
=====	=====	Prepost result combinations
resultCase(8)	-	1.0 x res(1) + 1.0 x res(4) + 1.0 x res(7)
resultCase(9)	-	1.0 x res(2) + 1.0 x res(5) + 1.0 x res(7)
resultCase(10)	-	1.0 x res(3) + 1.0 x res(6) + 1.0 x res(7)
resultCase(11)	-	2.0 x res(3) + 3.0 x res(6) + 4.0 x res(7)

6.1.1.5 Make the global load combinations – all combinations done in Presel



When all the superelements have been assembled to form the complete structure in Presel, the global load combinations (GLC) can be made. The Presel User manual shows how to assemble the structure and to make the combinations. In this case the global load combinations (GLC) are built up of basic loadcases (BL) as well as local load combinations (LLC) defined in GeniE.

The option for smart load combinations must be disabled so that the load combinations now become "true" basic loads and they will be part of the FEM file used by Presel when assembling the global model.

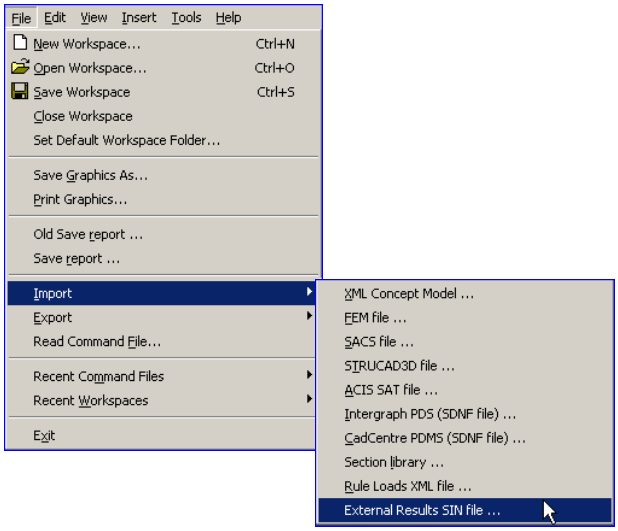
This alternative may be used in connection with wave load analysis.

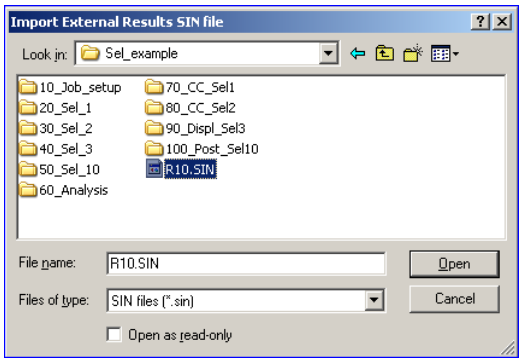
S.el. 10	S.el. 1		S.el. 2		S.el. 3	
Presel GLC	BL (LLC)	Content	BL (LLC)	Content	BL (LLC)	Content
-	BL1 (1)	Self-weight				
-	BL2 (2)	Equipments				
-	BL3 (3)	Line loads				
-			BL1 (4)	Self-weight		
-			BL2 (5)	Equipments		
-			BL3 (6)	Line loads		
-					BL1 (7)	Self-weight
1		1.0 x BL1 (1)		1.0 x BL1 (4)		1.0 x BL1 (7)
2		1.0 x BL2 (2)		1.0 x BL2 (5)		1.0 x BL1 (7)
3		1.0 x BL3 (3)		1.0 x BL3 (6)		1.0 x BL1 (7)
4		2.0 x BL3 (3)		3.0 x BL3 (6)		4.0 x BL1 (7)

As can be seen above there is consistency between the global load combinations on the highest superelement (s.el. 10) and the load case numbering on lowest level superelements (s.el. 1, 2 and 3). Also notice that any scaling of loads used in a load combination must be performed on the lowest level superelements. Typically, the global load combination consists of 1 x LLC11 from each of the lowest level superelements where LLC11 consist of 2 times the basic loadcase 3 in superelement 1.

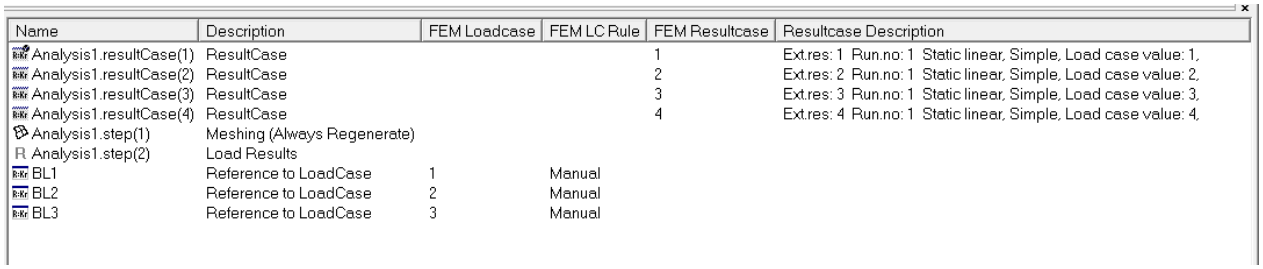
6.1.2 Post-processing

When the structural analysis has been performed it is possible to import the results back to each individual superelement. Open the relevant workspace for a superelement and import the results from ***File|Import|External Results SIN file***. In the example below the result file for superelement 10 - the R10.SIN file - is selected. The relevant results for a superelement will automatically be applied to the model.





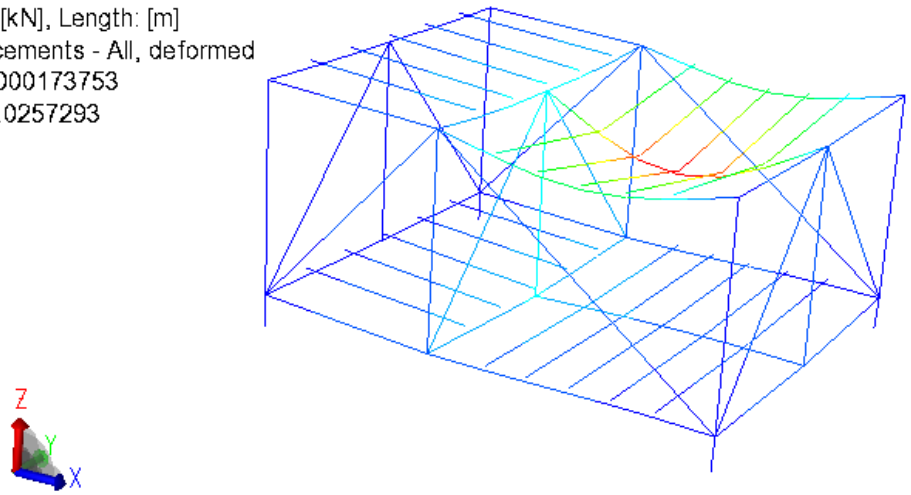
Result cases will appear in the browser after the SIN import.

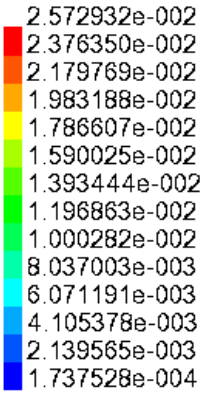


Name	Description	FEM Loadcase	FEM LC Rule	FEM Resultcase	Resultcase Description
Analysis1.resultCase(1)	ResultCase			1	Ext.res: 1 Run.no: 1 Static linear, Simple, Load case value: 1,
Analysis1.resultCase(2)	ResultCase			2	Ext.res: 2 Run.no: 1 Static linear, Simple, Load case value: 2,
Analysis1.resultCase(3)	ResultCase			3	Ext.res: 3 Run.no: 1 Static linear, Simple, Load case value: 3,
Analysis1.resultCase(4)	ResultCase			4	Ext.res: 4 Run.no: 1 Static linear, Simple, Load case value: 4,
Analysis1.step(1)	Meshing (Always Regenerate)				
Analysis1.step(2)	Load Results				
BL1	Reference to LoadCase	1	Manual		
BL2	Reference to LoadCase	2	Manual		
BL3	Reference to LoadCase	3	Manual		

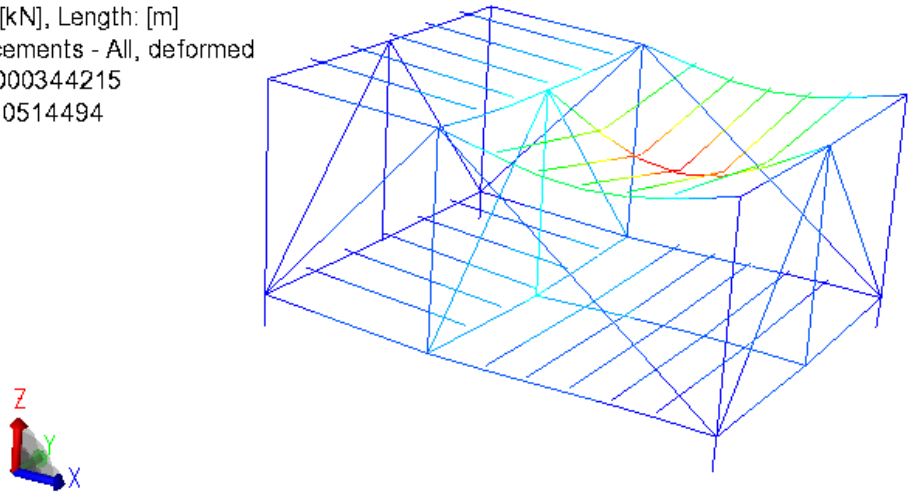
When importing results into GeniE, the FEM loads are imported and they can be verified, for example by looking at the result view. In the illustrations below, showing results for superelement 1, it can be observed that the displacements are twice as large in the resultcase 4 compared to resultcase 3. This is expected based on the numbers we were using earlier in this example; for resultcase 3 we put in 1.0 x BL3 and for resultcase 4 we put in 2.0 x BL3.

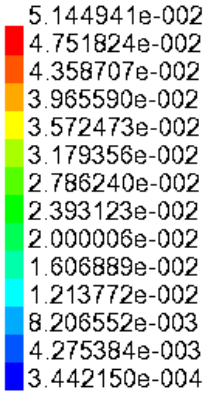
Analysis1
Analysis1.resultCase(3)
FEM Resultcase = 3
Force: [kN], Length: [m]
Displacements - All, deformed
Min: 0.000173753
Max: 0.0257293



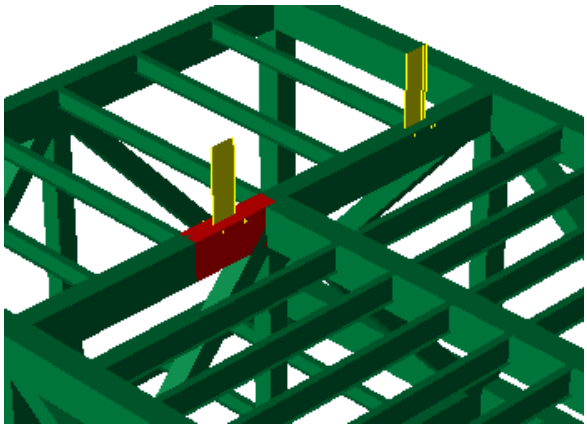


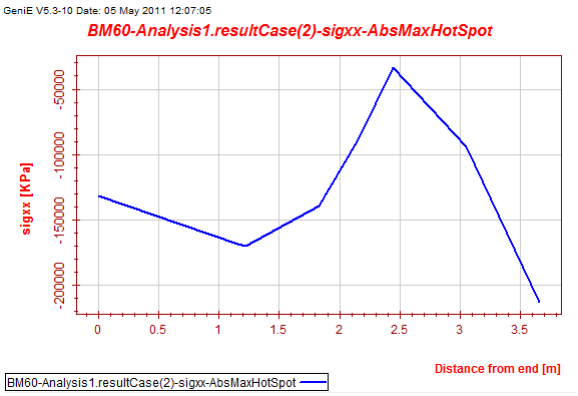
Analysis1
Analysis1.resultCase(4)
FEM Resultcase = 4
Force: [kN], Length: [m]
Displacements - All, deformed
Min: 0.000344215
Max: 0.0514494

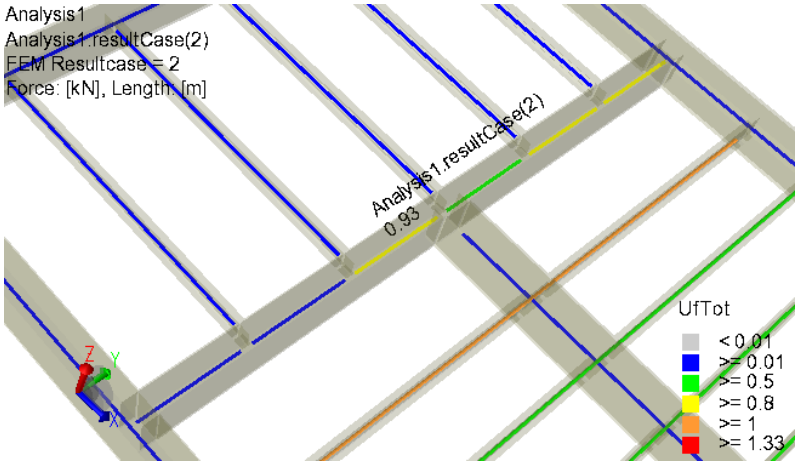


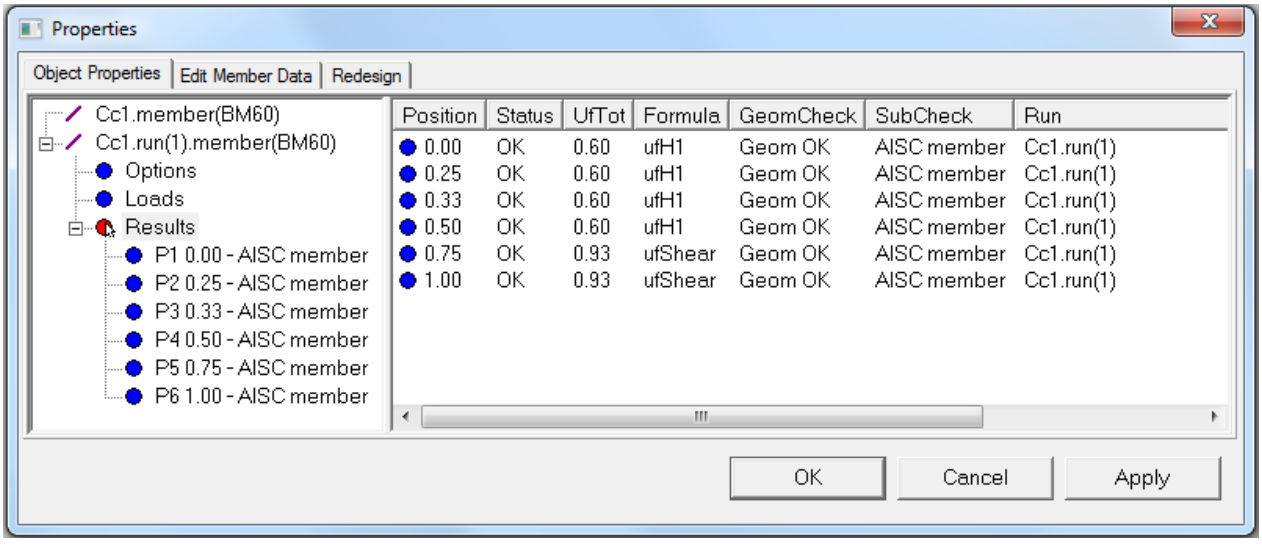


Post-processing can now be done including effects from local loads. The example below shows a detail for resultcase 2 on superelement 1 including a local load applied to one of the beams. In this case the local load effect influences the moment so that it has its peak outside one of the system pre-defined stress points along the beam (5 positions at 0, 0.25, 0.5, 0.75, 1.0).








Properties
Object Properties
Edit Member Data
Redesign
Cc1.member(BM60)
Cc1.run(1).member(BM60)
Options
Loads
Results
P1 0.00 - AISC member
P2 0.25 - AISC member
P3 0.33 - AISC member
P4 0.50 - AISC member
P5 0.75 - AISC member
P6 1.00 - AISC member
Position Status UfTot Formula GeomCheck SubCheck Run
0.00 OK 0.60 ufH1 Geom OK AISC member Cc1.run(1)
0.25 OK 0.60 ufH1 Geom OK AISC member Cc1.run(1)
0.33 OK 0.60 ufH1 Geom OK AISC member Cc1.run(1)
0.50 OK 0.60 ufH1 Geom OK AISC member Cc1.run(1)
0.75 OK 0.93 ufShear Geom OK AISC member Cc1.run(1)
1.00 OK 0.93 ufShear Geom OK AISC member Cc1.run(1)
OK
Cancel
Apply

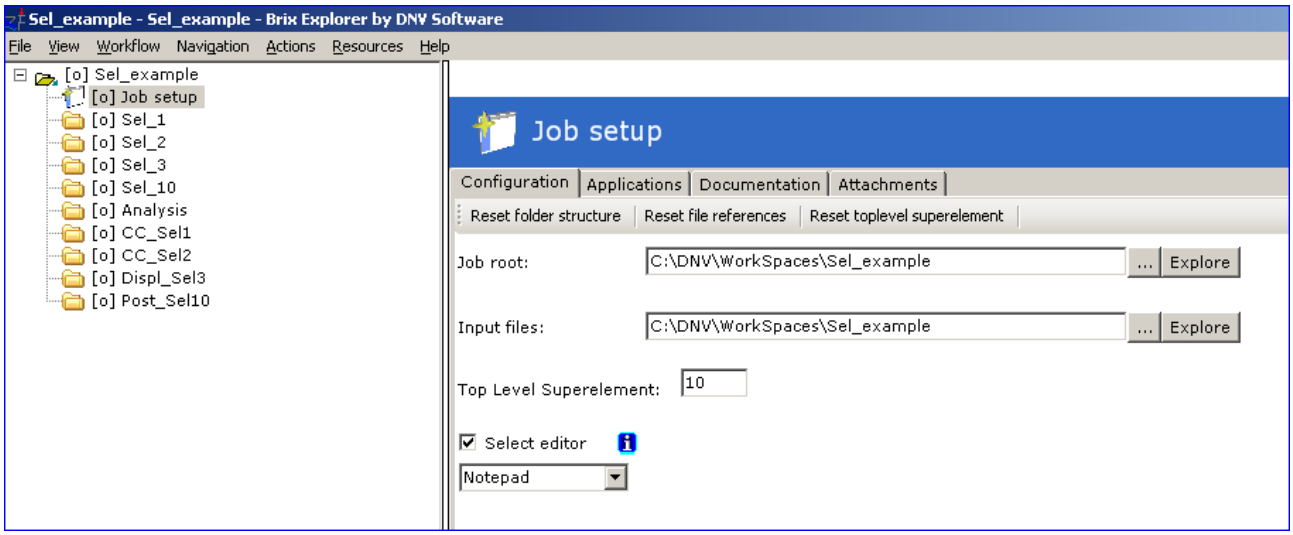
6.2 Process modelling in Brix Explorer

This chapter explains the steps used to build a process model in Brix Explorer. Main focus is on documenting which options that have been used – if you want to know in detail how to set up a process like this you can find this in the Brix User Manual.

There are several ways of making a process like this; the following example shows one way on how to do it.

6.2.1 Make the flowchart

The first step is to specify which tasks are to be carried out and in which sequence. After creating a new project in Brix Explorer you can define the tasks and which programs to use. The picture below shows the project "Sel_example" that consists of 10 tasks.



The first task is used to organise the design premises like program versions, reference documents, drawings, check-lists etc. For superelement analysis, the top level superelement number is also specified – in this case number 10.

The following tasks are used to specify the tasks to be done:

- Sel_1: Make superelement 1 in GeniE and export T1.FEM
- Sel_2: Make superelement 2 in GeniE and export T2.FEM
- Sel_3: Make superelement 3 in GeniE and export T3.FEM
- Sel_10: Make superelement 10, make load combinations and export T10.FEM (all in Presel)
- Analysis: Run structural analysis using Sestra (use system default input file)
- Prepost: Create result case combinations in Prepost (optional, required if only basic loadcases are taken up to the top level in Prepost)
- CC_Sel1: Import analysis results and perform code check on superelement 1
- CC_Sel2: Import analysis results and perform code check on superelement 2
- Displ_Sel3: Import analysis results and document displacements on superelement 3
- Post_Sel10: Look at displacements, forces and stresses of entire structure (T10) in Xtract

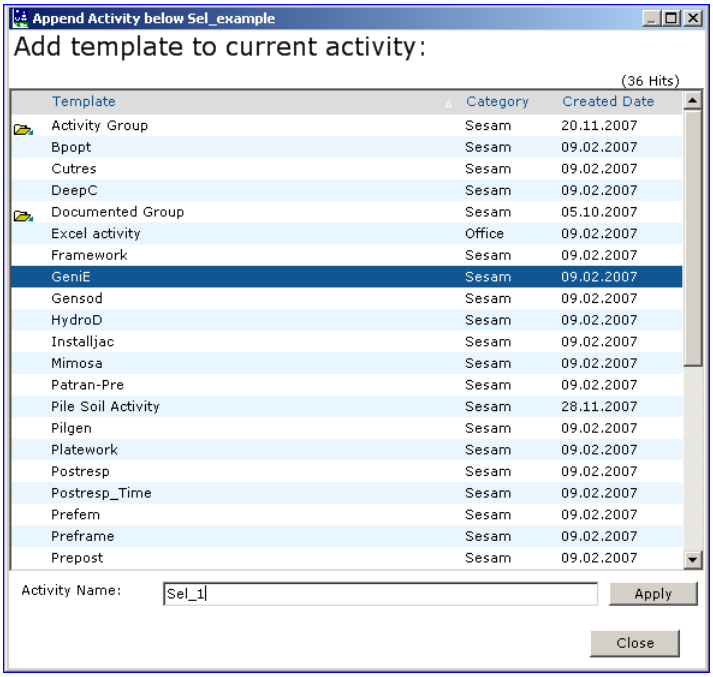
6.2.2 Execute the flowchart

Initially, the flowchart must be manually executed to specify the various option used in each task ("editing the controllers"). When all is set and you have generated the input file (or journal files) from manual sessions or by referring to input files (or journal files) the flowchart may be executed automatically by using the "Run Hierarchy" option. All file handling is automatically taken care of by Brix Explorer.

In the following, the manual execution of the flowchart is focused.

6.2.2.1 Run GeniE

Define the task name and refer to GeniE as program to use.



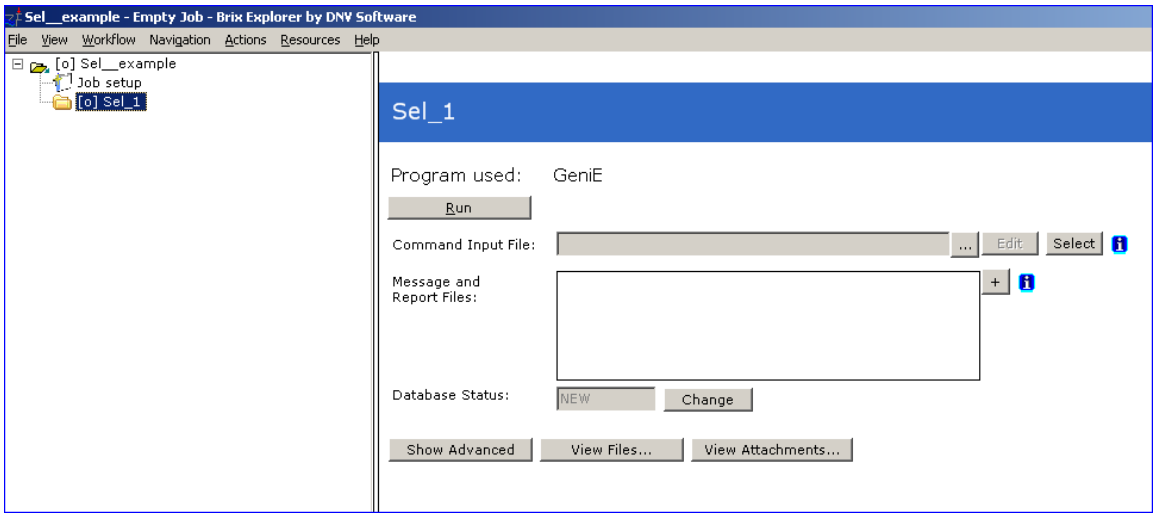
GeniE is started by clicking the Run button. In this case no js-file is automatically imported when running the program.

You may refer to a js-file that always is imported when you specify the database status to *New*.

The status is automatically modified to Old when you have exited GeniE.

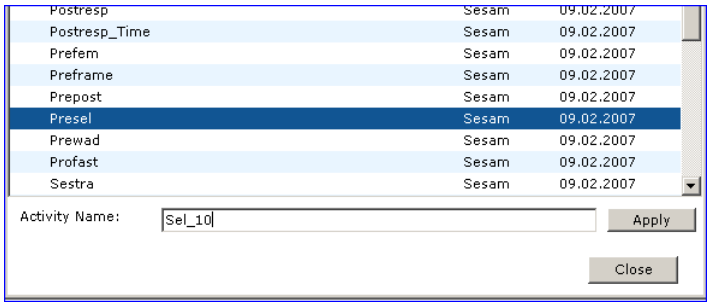
You do not need to use the Advanced View.

This and similar activities are used to define superelements T1, T2 and T3.

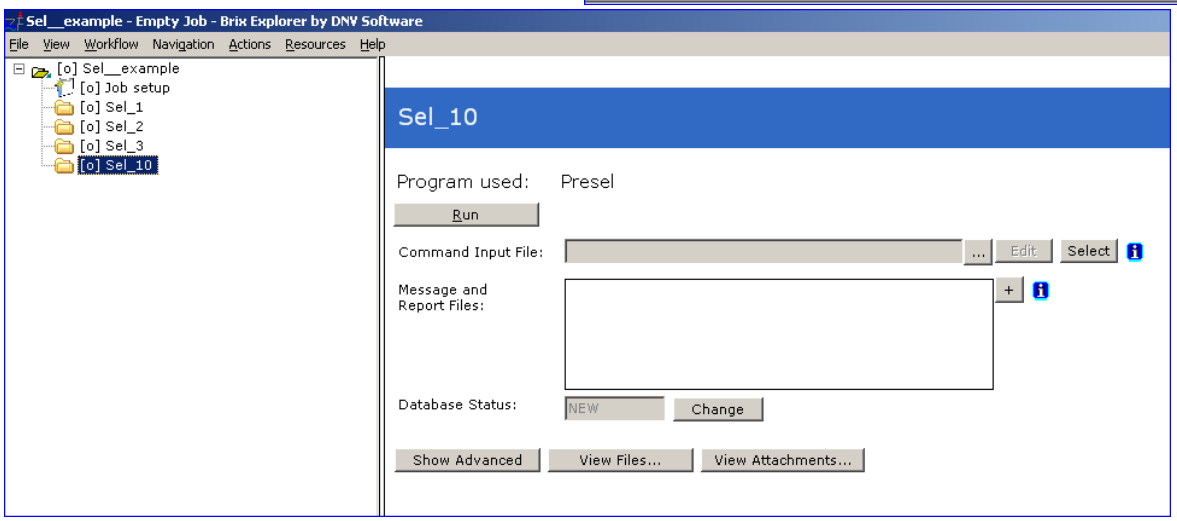


6.2.2.2 Run Presel

Define the task name and select Presel as program to use.



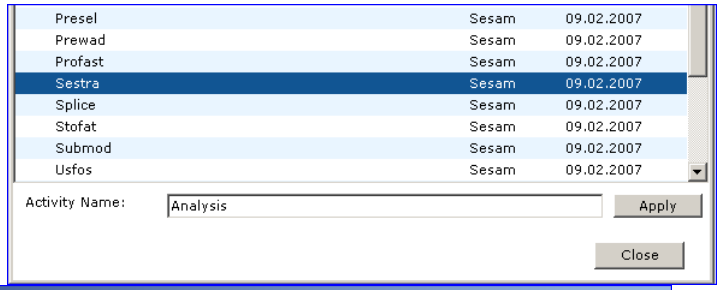
Presel is started by clicking the Run button and like for GeniE you may do manual modelling or refer to an existing input file.

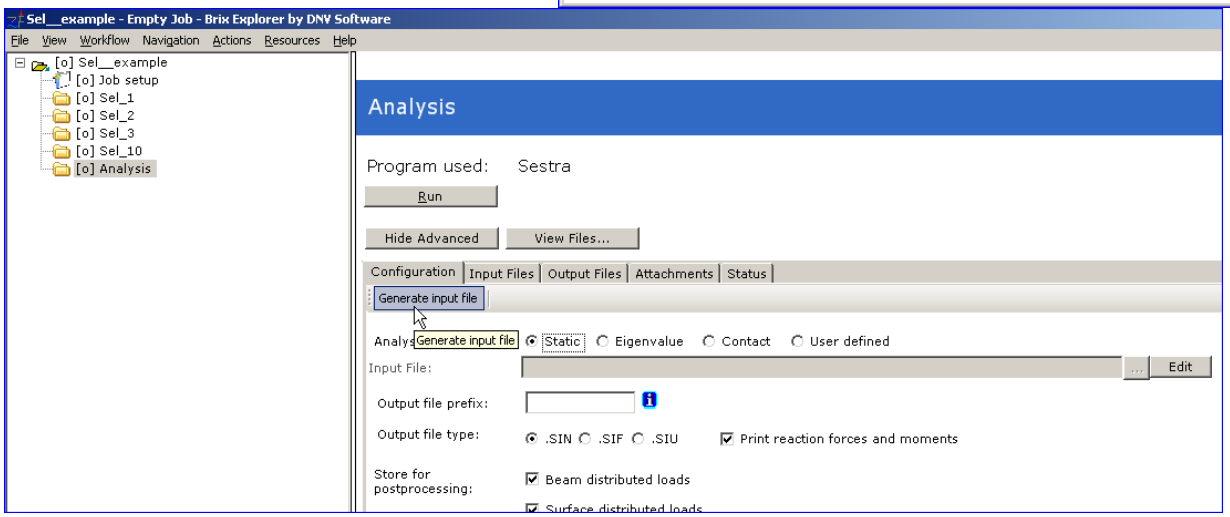


6.2.2.3 Run Sestra

Define the task name and select Sestra as program to use.

In this case you need to open the "Advanced view" assuming there are no user defined input file from before. Specify static analysis run, click on generate file and start Sestra by clicking the Run button.





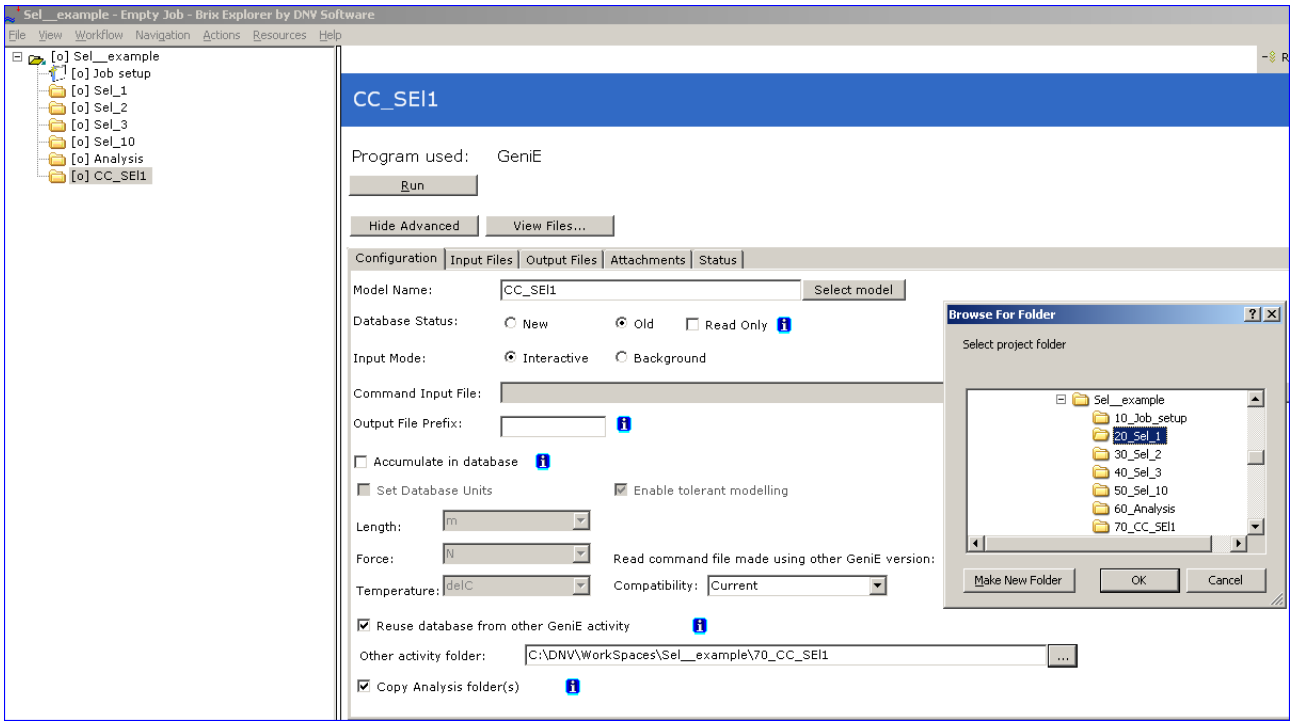
6.2.2.4 Run Prepost

Result case combinations can also be created in Prepost. This is optional and is only required if basic loadcases are taken up to the top level in Prepost

6.2.2.5 Re-use a GeniE activity to do post processing

In this case we want to re-use an existing GeniE model in a new activity when importing results from the superelement analysis to do code checking.

Define the task and specify GeniE as the program to use. In this case it is necessary to view the advanced view to set the options to do the task. Code checking of superelement 1 is performed using the model defined in task *Sel_1* and then results as computed in the task *Analysis*.



1) Set the database status to *Old*.

2) Click on "*Reuse database from other GeniE activity*".

3) Locate the relevant activity folder – in this case 20_sel_1.

4) Click on "*Copy Analysis Folder*" to ensure that all the analysis details are transferred to the new task.

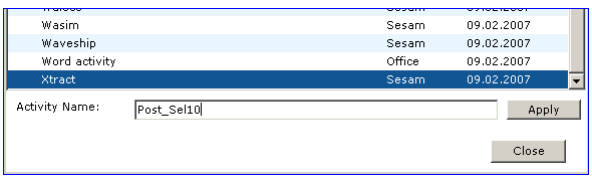
5) Click *Run* and do the necessary code checking and other post-processing.

6) When the task is completed it is advised to set database status to "*Read Only*" – you may now later open the task again to investigate the model, results, make reports or pictures. In case you accidentally modify the model, this is not saved.

The similar steps can be used to do the tasks *CC_Sel2* and *Displ_Sel3*.

6.2.2.6 Run Xtract

The final step is to specify the task to do post-processing of the entire structure (T10) based on Xtract.



The program is started like shown in previous tasks and it will automatically pick up the highest superelement.

7. EXECUTION OF GENIE

This Chapter describes which files are used by the program in connection with code checking of beams.

7.1 Program Environment

See the GeniE User Manual Vol. I for information on the hardware requirements of GeniE and how to execute the program.

7.2 Files created during code checking

GeniE will create the following files during code checking and reporting of such:

- The journal file format *.js
- The database, format *.gni
- The XML Concept Model file *.xml
- The GeniE saved report, format *.lis, *.html, *.xml (for Excel and Word)
- The GeniE saved graphics file, format *.gif, *.jpg, *.eps, *.bmp, *.tga, *.tif, *.ps, *.vrml, *.dfile
- The GeniE_log.txt file in case of model errors and/or warnings

Notice that the ***File|Export|GeniE Journal File*** (the "clean" journal file) does not contain code checking definitions nor any information pertaining to shell models (curved plates). For a neutral storage of the model, you should use the ***File|Export|XML Concept Model file***. This file can be imported to new workspaces.

This page is blank for printing

8. DIALOG DESCRIPTION

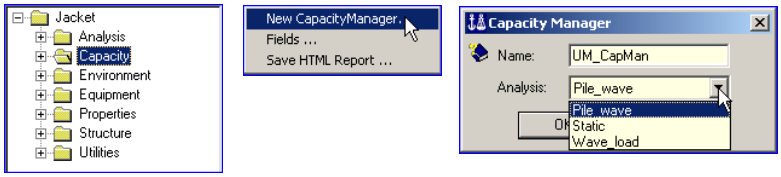
This Chapter lists the relevant dialogs used in connection with code checking of beams. For all other dialog definitions reference is made to GeniE User Manual Vol. I.

All dialogs relevant for code checking are available from the browser and partially from the graphical window by using the context sensitive menu (select objects and right click to access these menus).

8.1 Code checking menus from the browser

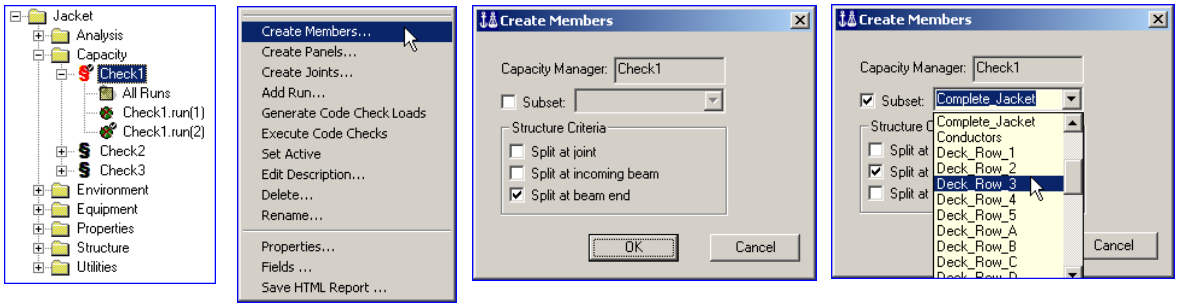
8.1.1.1 Make a new capacity manager

To decide which analysis to include in the code check runs.



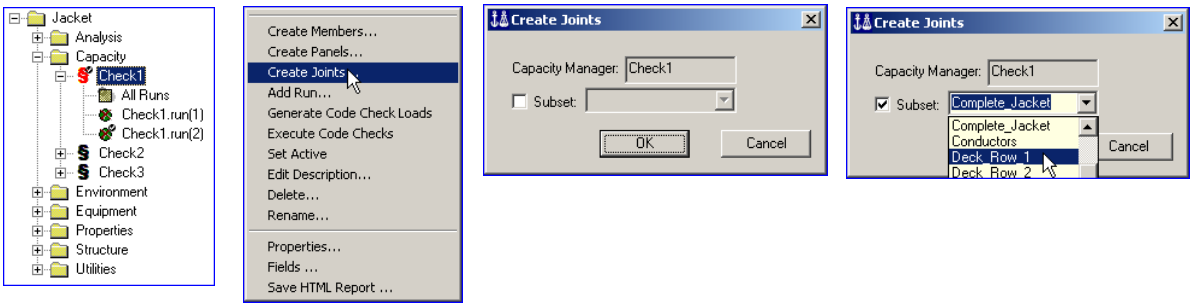
8.1.1.2 Make capacity members

To make a capacity model and decide which parts of the structure to include in the code check run. The default buckling lengths are also decided based on how a beam is split.



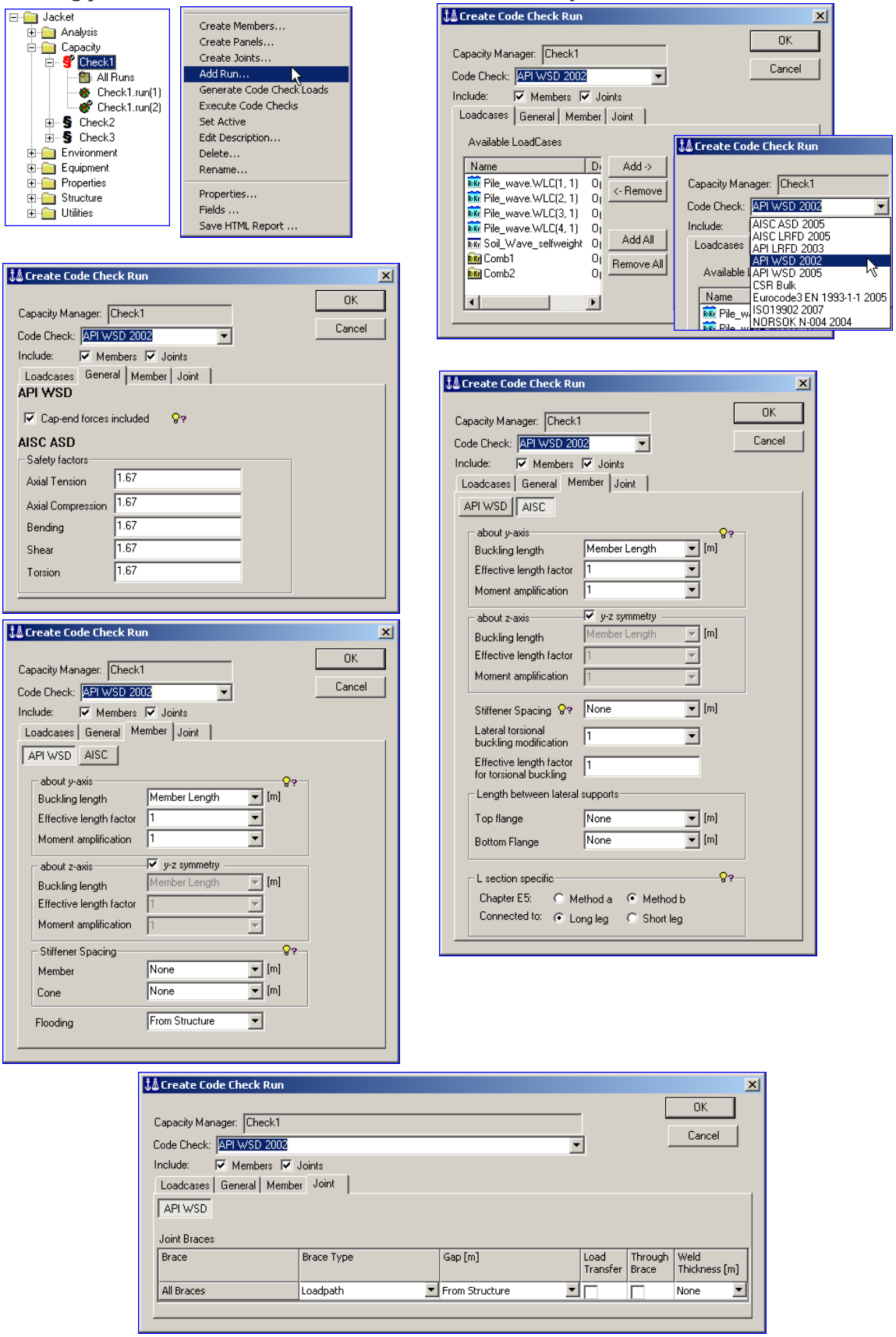
8.1.1.3 Create joints

To create joints to be part of a punching shear check and to decide which parts of the structure to include in the code check run.



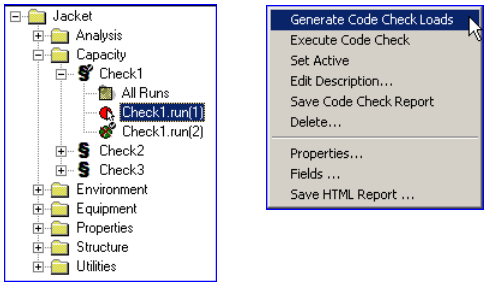
8.1.1.4 Add a run

To decide which code check standard to use, to specify which load cases to use and to set up the global code checking parameters for the code check, the members and the joints.



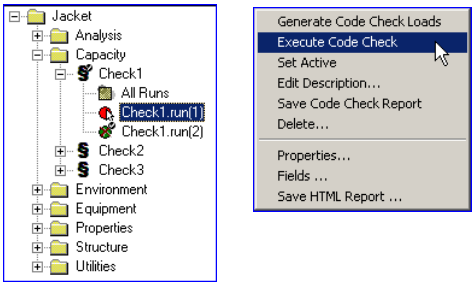
8.1.1.5 Generate code checking forces

To compute the code checking positions and forces to be used in the code check. If you do it from the "All Runs" folder the operation is performed for all the code check runs you have defined.



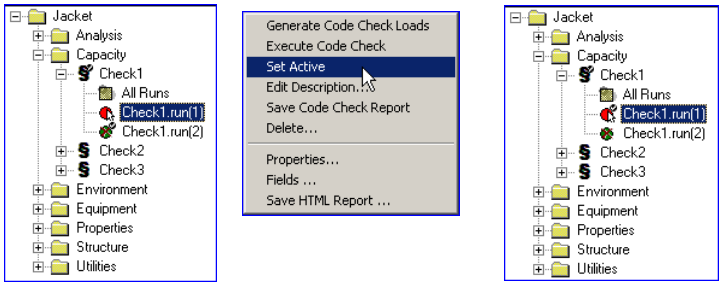
8.1.1.6 Execute the code check

To perform the code check. All code check runs will be executed if you do it from the "All Runs" folder.



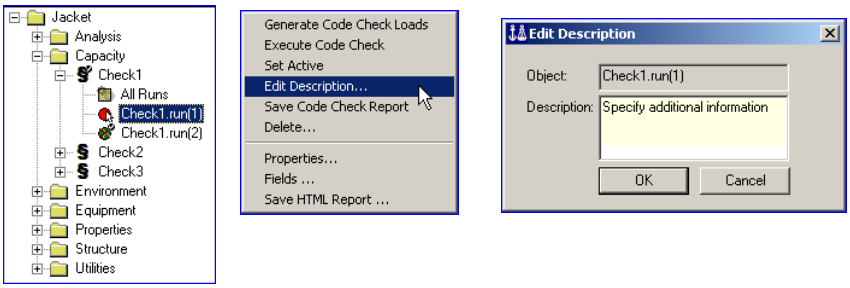
8.1.1.7 Set a code check run to active

If you have multiple code check runs, you specify which one is active from this command. Notice the default symbol in the browser.



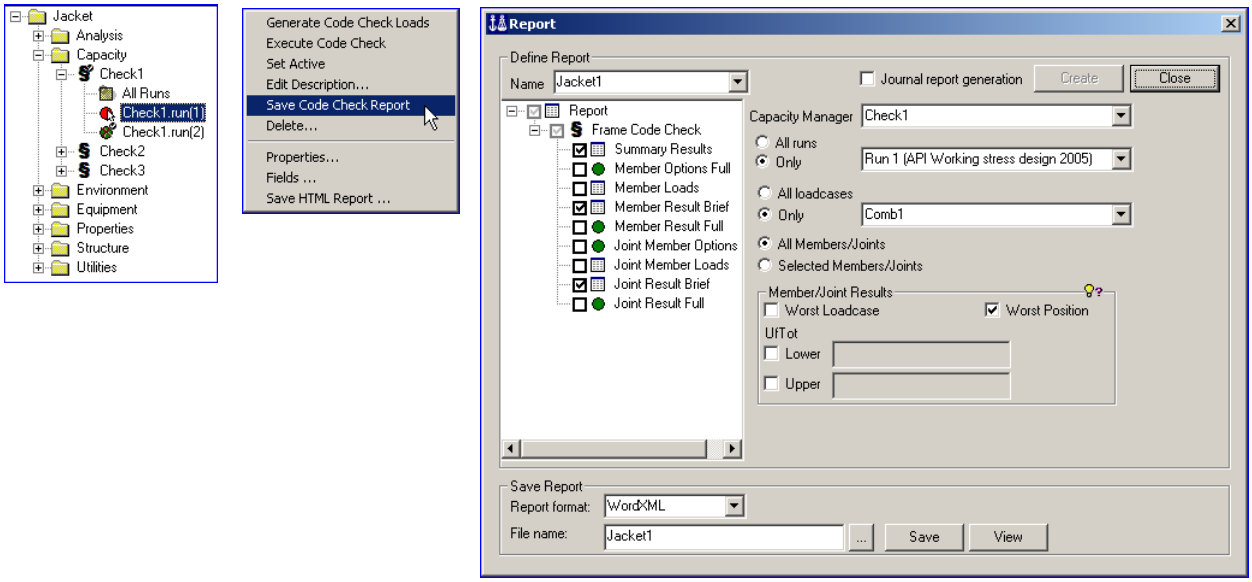
8.1.1.8 Edit a code check description

Include additional information to a code check run (or a capacity manager). The information is listed in the report.



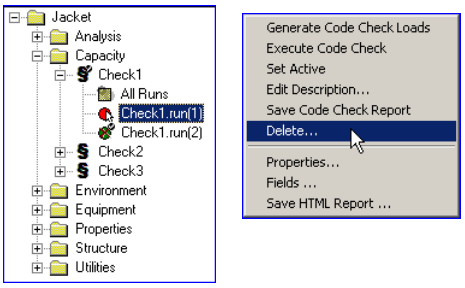
8.1.1.9 Save a code check report

This command will save a code check report as described in Section 3.9 *Make a report*. The only difference is that the report dialog assumes that a frame code check report will be made – hence the other chapters for structure, loads and so on are not added to the report content (you may add these manually).



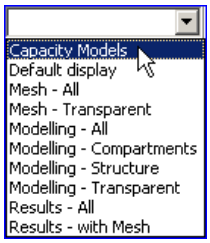
8.1.1.10 Delete

This command deletes the code check run from the capacity manager.



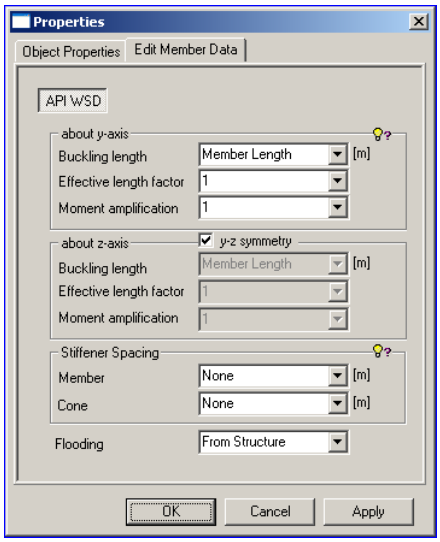
8.2 Code checking menus from the graphical window

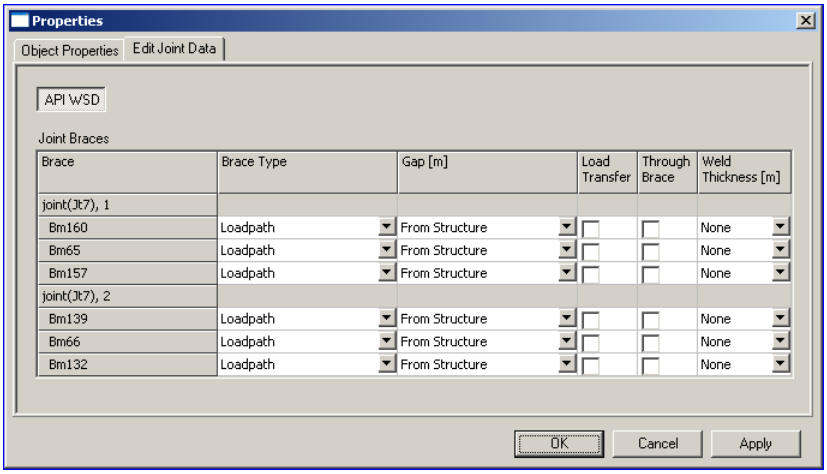
Notice that you need to select a capacity object (member or joint) and right click to see the context sensitive menu. You should thus use the pre-defined view setting Capacity Models when viewing your capacity model.



8.2.1.1 Properties

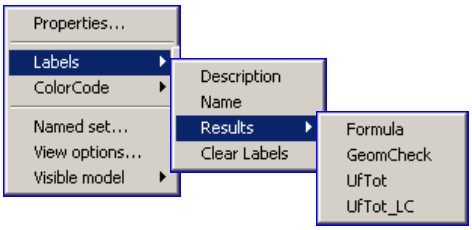
Specify capacity parameters that apply to the selected member or joint only. These settings will over-ride the system default and other global settings specified when you define the code check run. Notice that the content of the property dialog depends on capacity member or joint as well as which code check has been selected. The examples below are based on API WSD.





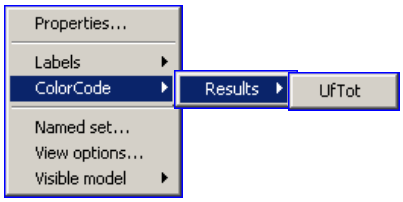
8.2.1.2 Labels

When the code check has been executed you may add labels as shown below to the capacity members. Notice that "Clear Labels" will remove the labels on the selected object(s) only. If you want to remove all labels for the whole model you should use the exclamation mark.



8.2.1.3 Colour coding the results

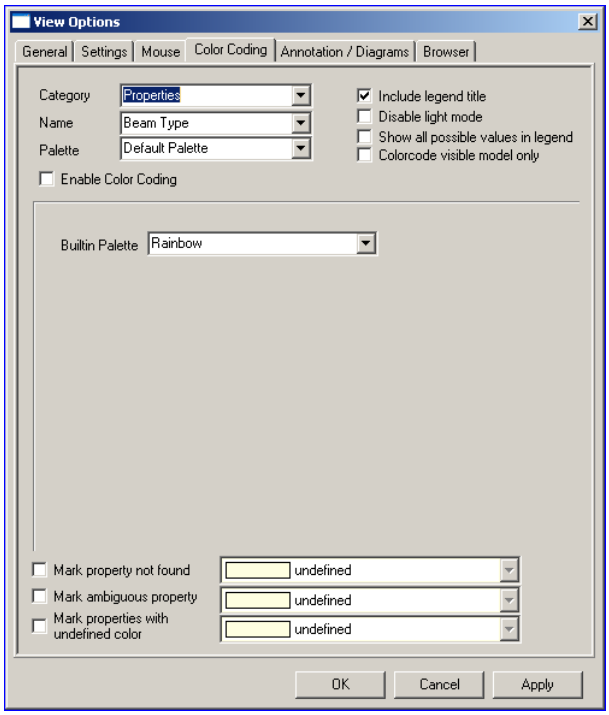
You can colour code the code checking results by using this command. To customize the colours, number of levels and the thresholds see explanation in next Section.



8.3 Code checking menus from the pulldown menu

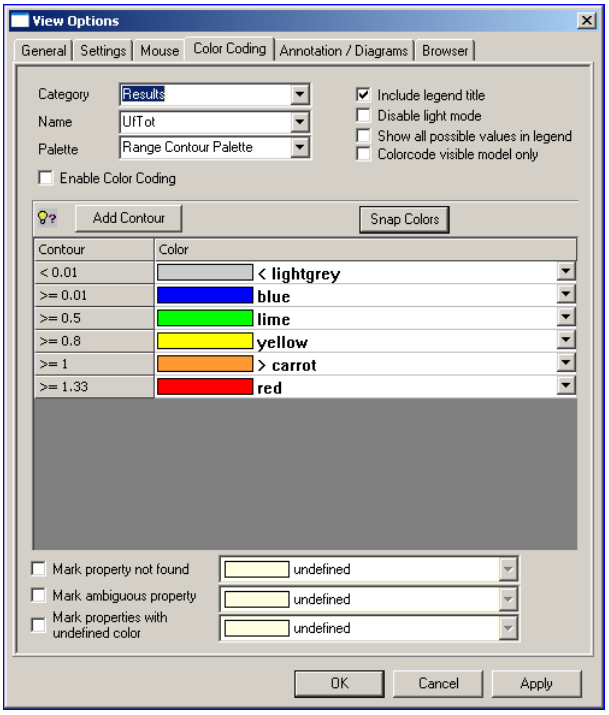
8.3.1.1 Specify the colour coding of code check results

The pulldown menu ***View|Options|Color Coding*** is used to customise the colours, number of levels and the thresholds when colour coding code checking results.

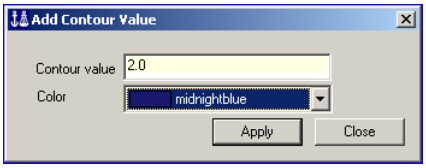


The view options dialog is used to customize how you want to view your model in various modes – typically during modelling, load application, analysis or code checking stage.

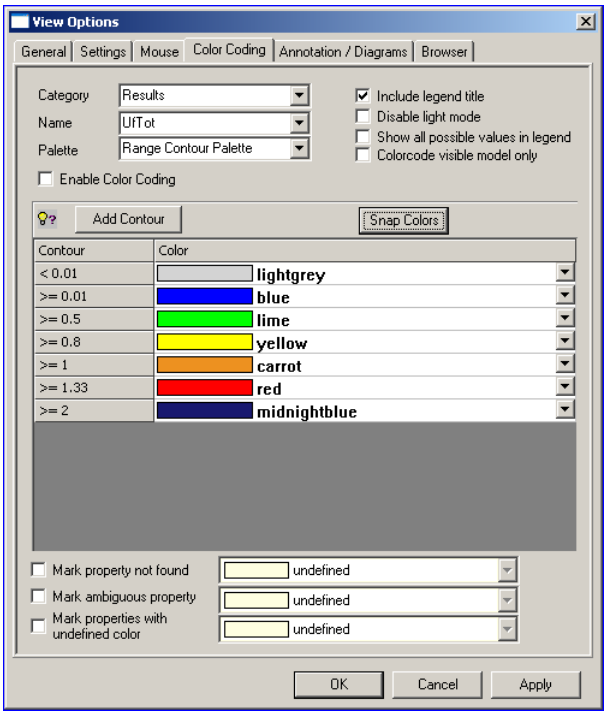
If you select "Category" to "Results" as shown below you can modify number of contour levels, the thresholds and the colours within each range.



Additional levels are added from "Add Contour".



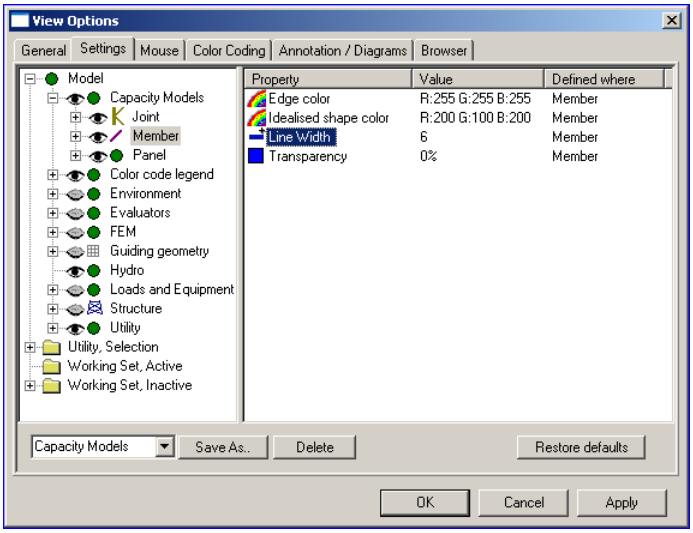
Contours are removed by selecting a contour from the contour column, RMB and *Remove Contour*.

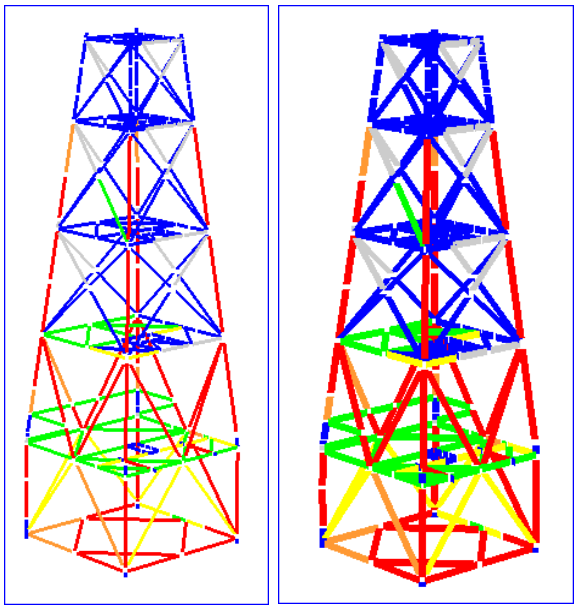


8.3.1.2 Specify the line width of capacity members

You may change the line width of the capacity members for better viewing graphically or on pictures. The pictures below shows a picture generated by default line width (3) and a picture with line width 6.

The line width is modified from the pulldown menu ***View|Options|Settings*** (remember to specify which view you want to modify, in this case "Capacity Models").





This page is blank for printing

9. APPENDIX A: REFERENCES

1. API/WSD 2002 – The implementation of API WSD is according to "Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms—Working Stress Design". API RECOMMENDED PRACTICE 2A-WSD (RP 2A-WSD). TWENTY-FIRST EDITION, DECEMBER 2000. ERRATA AND SUPPLEMENT 1, DECEMBER 2002
2. API/WSD 2005 - The implementation of API WSD is according to "Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms—Working Stress Design". API RECOMMENDED PRACTICE 2A-WSD (RP 2A-WSD). TWENTY-FIRST EDITION, DECEMBER 2000. ERRATA AND SUPPLEMENT 2, OCTOBER 2005.
   The joint capacity check also includes error/misprint corrections in ERRATA 3, January 2007
3. API LRFD 2003. The implementation of API LRFD is according to "Planning, Designing and Constructing Fixed Offshore Platforms—Load and Resistance Factor Design". The implementation of API LRFD 2003 is according to the revision "1st Edition / July 1, 1993 / Reaffirmed, May 16, 2003".
4. NORSOK N004 – The implementation of Norsok N-004 is according to NORSOK STANDARD N-004, Rev. 2, October 2004, Design of steel structures
5. ISO 19902. The implementation of ISO is according to "International Standard ISO 19902, Petroleum and natural gas industries – fixed offshore structures" 1st Edition, 1 December 2007.
6. AISC. The implementation of AISC is according to "ANSI/AISC 360-05 An American National Standard; Specification for Structural Steel Buildings", March 9, 2005. The implementation of AISC is according to the revision March 9, 2005. The check covers design/utilisation of members according to the provisions for *Load and Resistance Factor Design* (LRFD) or to the provisions for *Allowable Strength Design* (ASD).
7. EUROCODE 3 - The implementation of Eurocode 3, EN 1993-1-1 is according to Eurocode 3: Design of steel structures – EN 1993 Part 1-1: General rules and rules for buildings, 2005

This page is blank for printing

10. APPENDIX B: CODE CHECKS AND NOMENCLATURE

This Appendix describes which checks are performed and the nomenclature used.

For reference to how the various standards are implemented, please see separate documents available from GeniE's help menu.

10.1 The checks performed

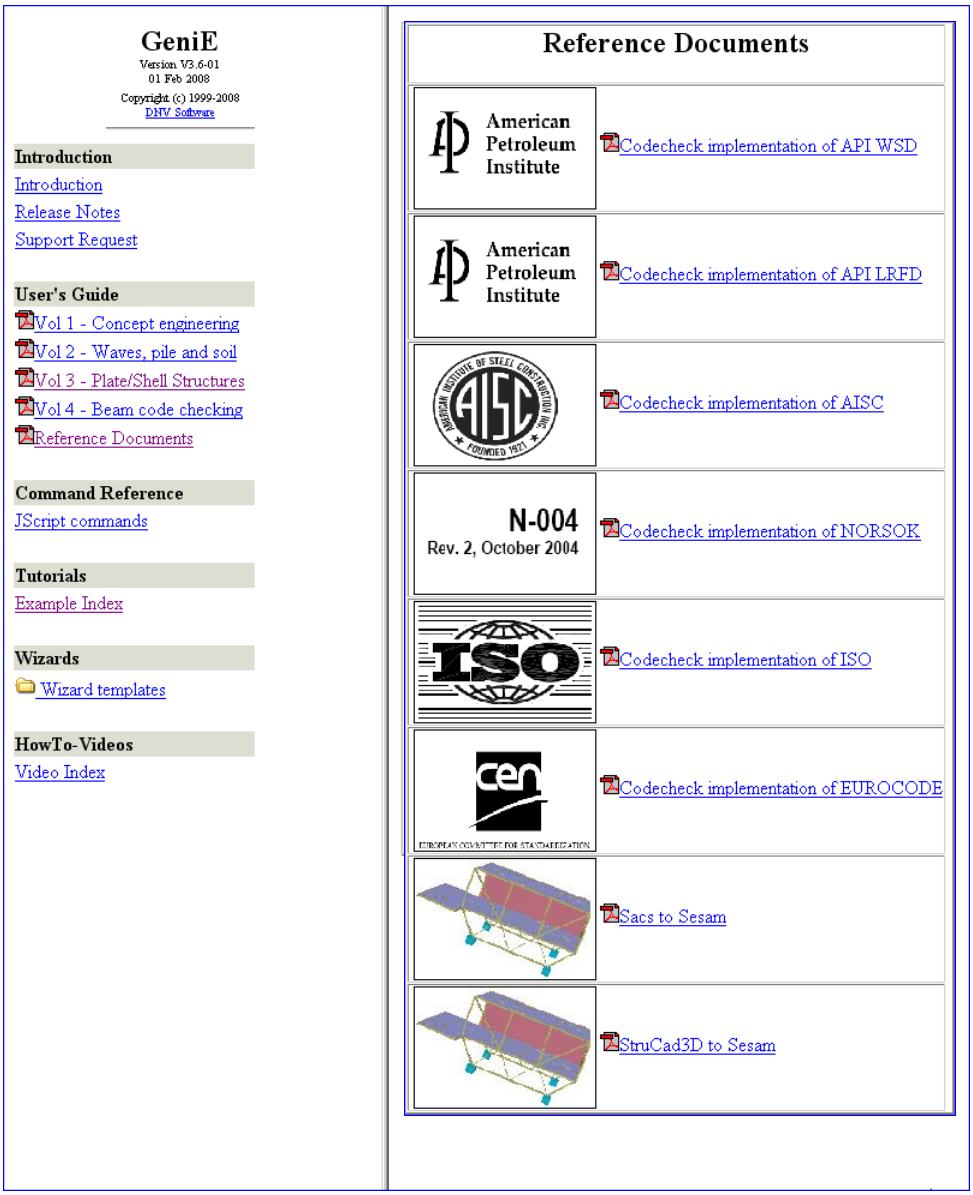
GeniE will perform two checks during the code check execution run:

1 To find utilisation factors
   - Formulas in codes describe design strength of members and joints
   - Failure modes:
     - Strength of cross section
     - Stability of member due to compression and/or moments
     - Hydrostatic collapse
     - Punching shear of chord (can) from incoming braces (stubs) in a tubular joint
     - Conical transition (tubular)
   - Formulas converted into calculation of interaction ratios in GeniE based on actual loading and design strength
     - Interaction ratio < 1.0 then OK
     - Interaction ratio > 1.0 then failure (require re-design)

2 To perform a geometry check according to the criteria specified in the code of practice. Typically for API/AISC these are
   - API (failure when below criteria are exceeded)
     - D/thk < 300
     - Thk ≥ 0.25 inch (6 mm)
   - AISC (failure when below criteria are exceeded)
     - slenderness L/r < 300 (member in tension)
     - slenderness L/r < 200 (member in compression)
     - D/t < 273 (pipe section only)
       - < Izc/Iz < 0.9 (I section only)
     - h/tw ratio according to Section F13.2. Reported relative to variable limit.
     - Aweb/Acf < 10 (according to Section F13.2)
     - Leg length ratio < 1.7 (L section only)

10.2 Nomenclature

The nomenclature for each of the code checks may be found from GeniE's help pages under "Reference Documents".



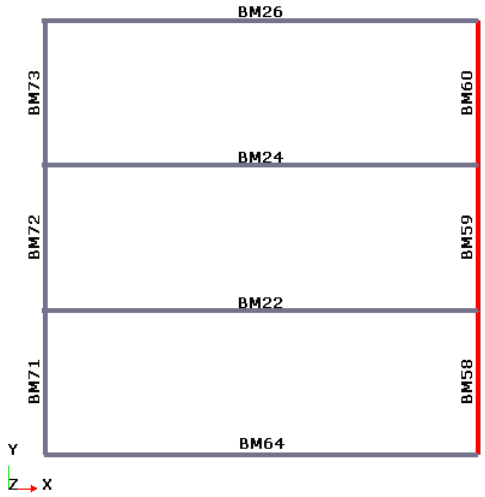
11. APPENDIX C: CODE CHECKING BASED ON FRAMEWORK

This appendix describes how to do code checking of beams using the program Framework. You should use this approach if you want to use older versions of the code checks than implemented in GeniE.

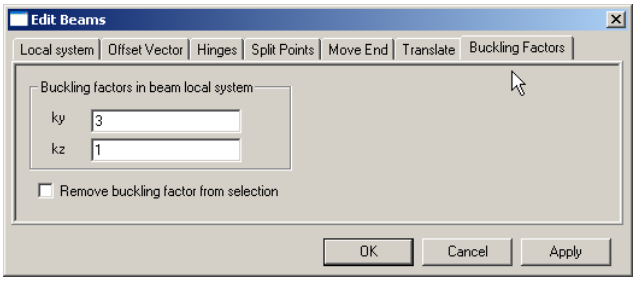
The code checking is done in the program Framework. The program may be started from GeniE. When model data is imported in Framework, all concept information is read in. Examples of concept data may be names of objects (beams, sections, materials, load case names, joint names, buckling length factors) and relationship between beam members and the finite elements.

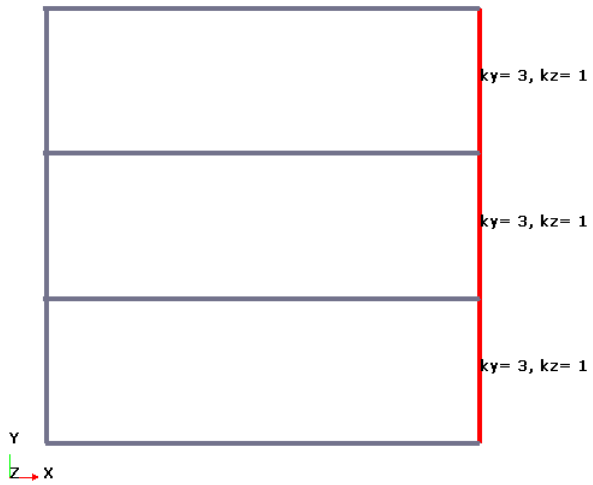
This means that when referring to an object the name in GeniE is used, or you may use the finite element numbering system if this is the preferred solution. Note that when creating a model in GeniE the number of characters in a name should not exceed eight (8) to avoid truncation of names.

Prior to code checking you may want to add buckling length factors to the beams. The buckling length factors are being used by Framework to calculate the beam buckling length when performing the code checks. In the example below beams BM58, BM59, BM60 are given buckling factors to simulate buckling in local y-direction covering the span from beam BM64 to beam BM26. Similarly, the buckling length in local z-direction shall be equal to the beam length.



Select the beams, force the context sensitive menu and choose Edit Beam and Tab Buckling Factors



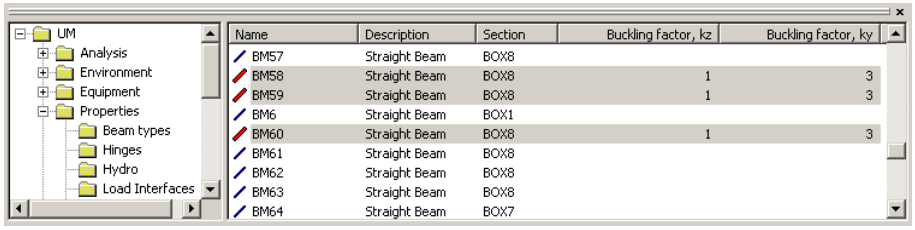


The buckling factors are defined and applied to the selected beams.

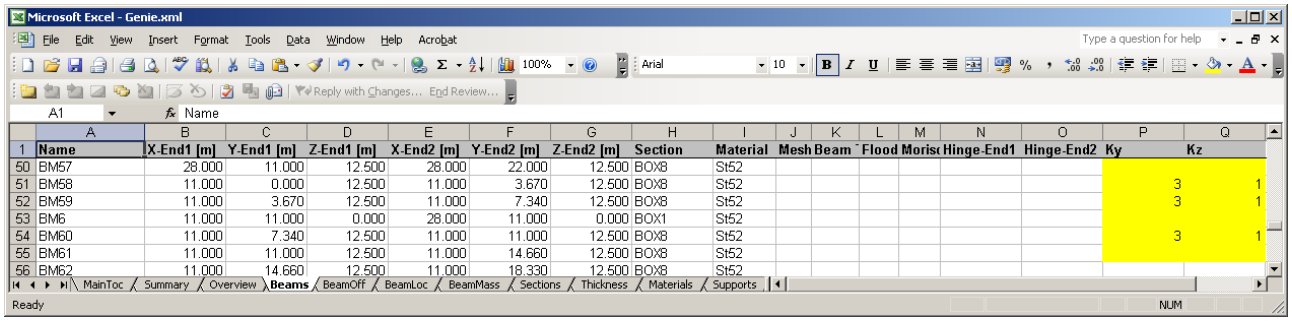
Note that if you split a beam or join beams, the assigned buckling factors will be set to default (k_y=-1 and K_z=-1) which means default buckling factors will be used by Framework).

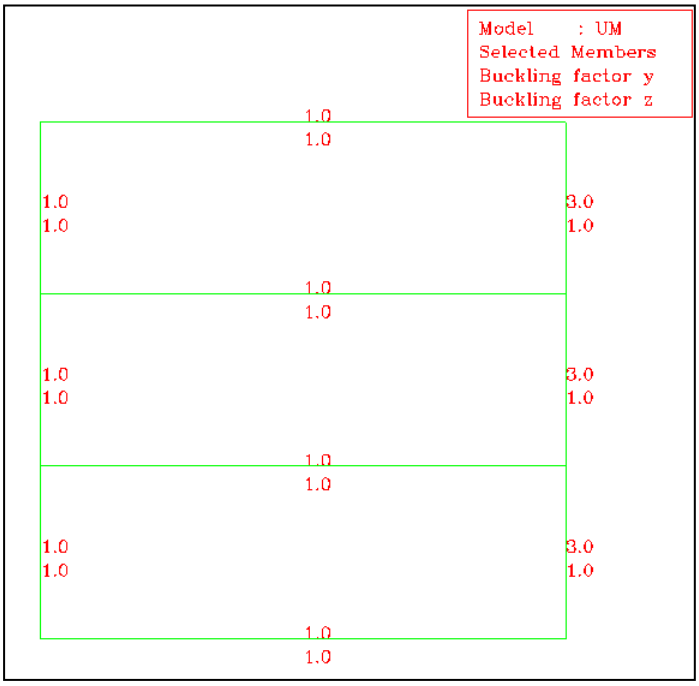
The default values are not shown when you label the buckling factors.

You may also verify the applied buckling factors from the browser (by adding the necessary fields to the structure browser), the saved report or from Framework itself. All options are showed below.



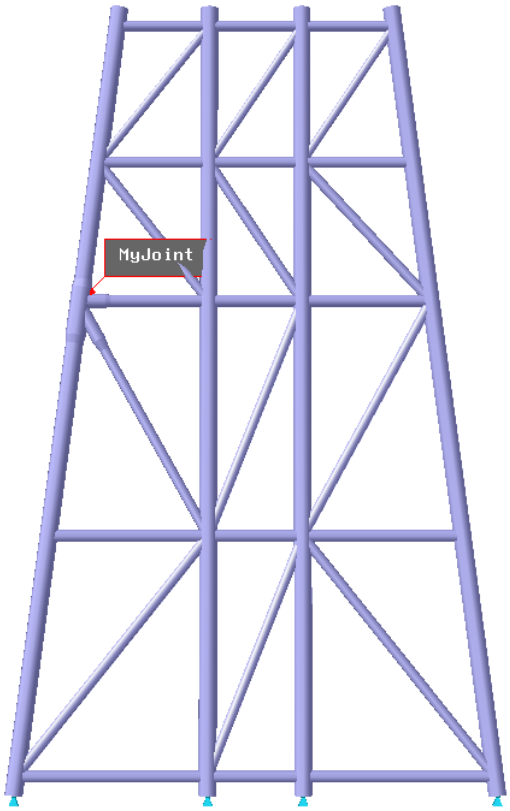
The fields kz, and ky are added to the browser view.





The buckling factors as reported by Framework. See below on how to start the code checking.

The code checking program is started from ***Tools|Analysis|Frame Code Check***. This Chapter focus how to set up the code checking model and how to control which data are exported to Framework. For more details on how to do the actual code checking (yield, buckling, punching), fatigue (deterministic, stochastic), or earthquake analysis, reference is made to the Framework User Manual.

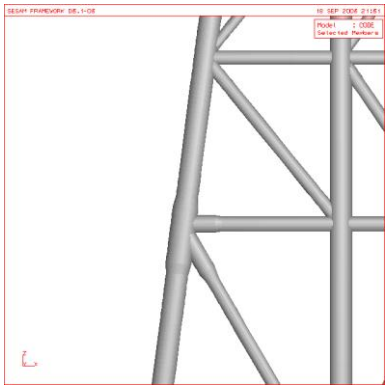


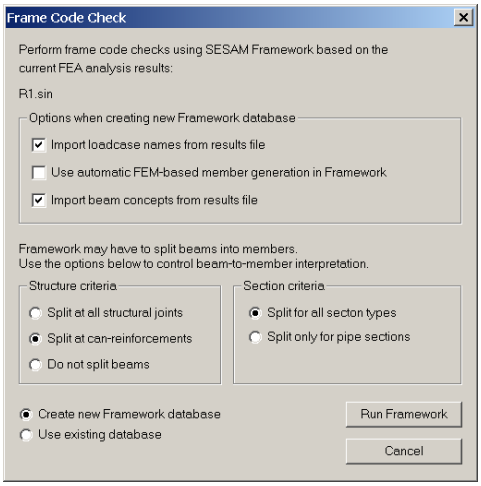
To show how a code checking model can be set up by GeniE, the jacket frame to the left is used.

This model consists of tubular members only, and there is one joint where cans, stubs, cones, and gaps are assigned. This structural joint is named Myjoint.

To visualise only the joint and incoming members, select the joint, RMB, *Select Connected Beams* and show these only.

The picture at right shows the model imported to Framework

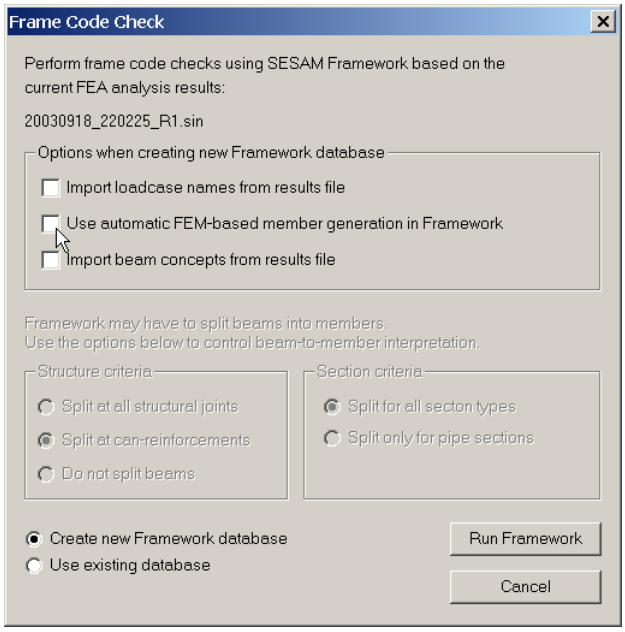



Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

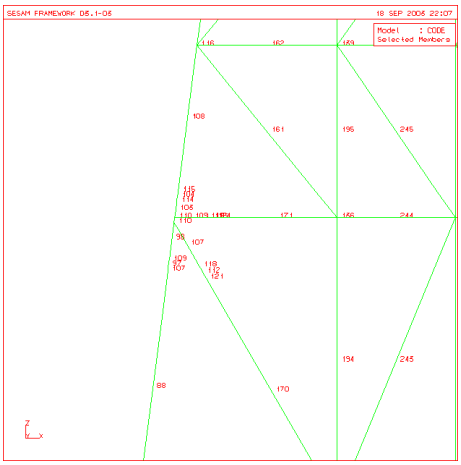
To set up the model for code checking, the dialog sheet at left is used (this is activated from ***Tools*|*Analysis*|*Frame Code Check***)

In the following, the options on how to split up the structure to create wanted member lengths are discussed.

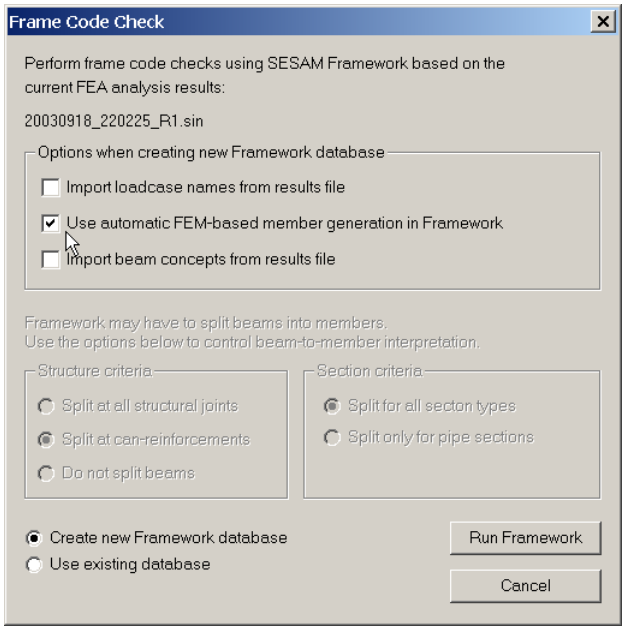
11.1 Import the model using finite element numbering


Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
20030918_220225_R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

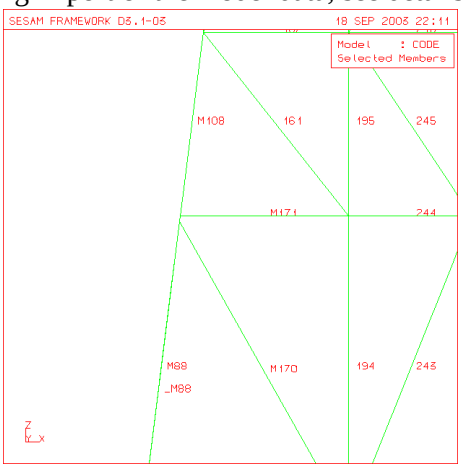
No concept information is exported to Framework. To do code checking all references must be made to finite element and node numbering. These numbers are identical in GeniE and Framework. Below is an example of the element numbers close to the structural joint Myjoint.



11.2 Generate members in Framework


Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
20030918_220225_R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

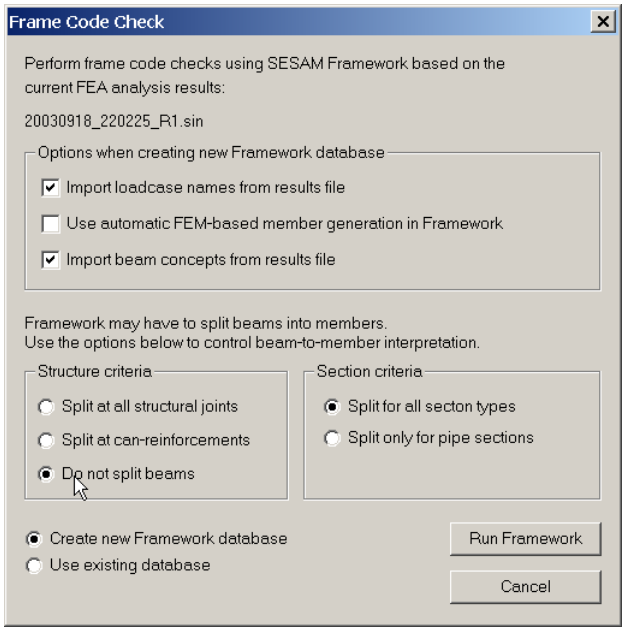
In Framework it is possible to automatically create members between two structural joints where each member may consist of many finite elements. By activating this feature, Framework will do this operation during import of the model data, see details close to Myjoint.



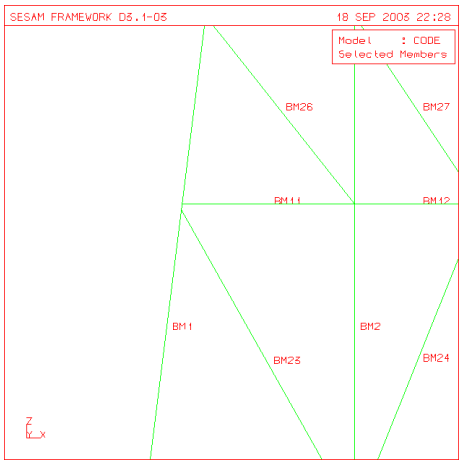
Both examples above imply that the user has knowledge about the finite element model, and as such not utilising the high level concept information already built into the model. By using the automatic member creation, the buckling length is always set equal to the member length unless manually specified or automatically computed by the program using an eigenvalue analysis approach (tubular members only).

Since members are modelled independently of finite element numbers in GeniE, the default buckling lengths are more inline with the real situation as compared to a finite element modelling approach.

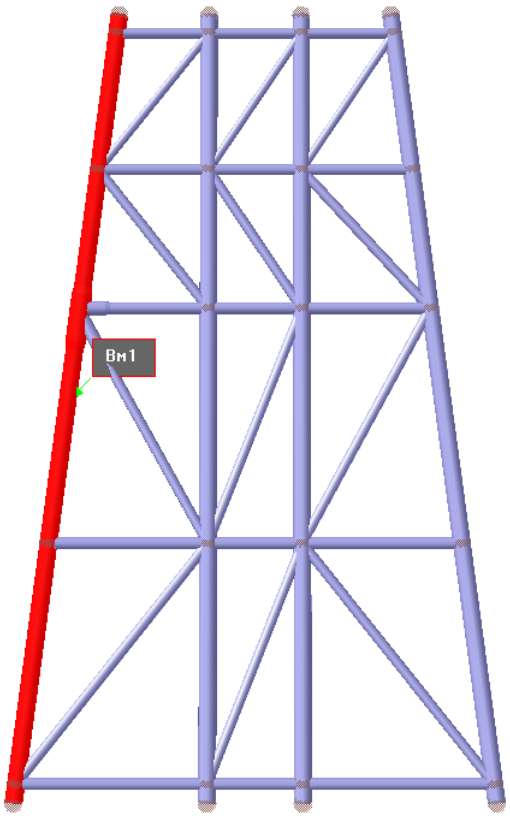
11.3 Import the model using concept names, no split of structure

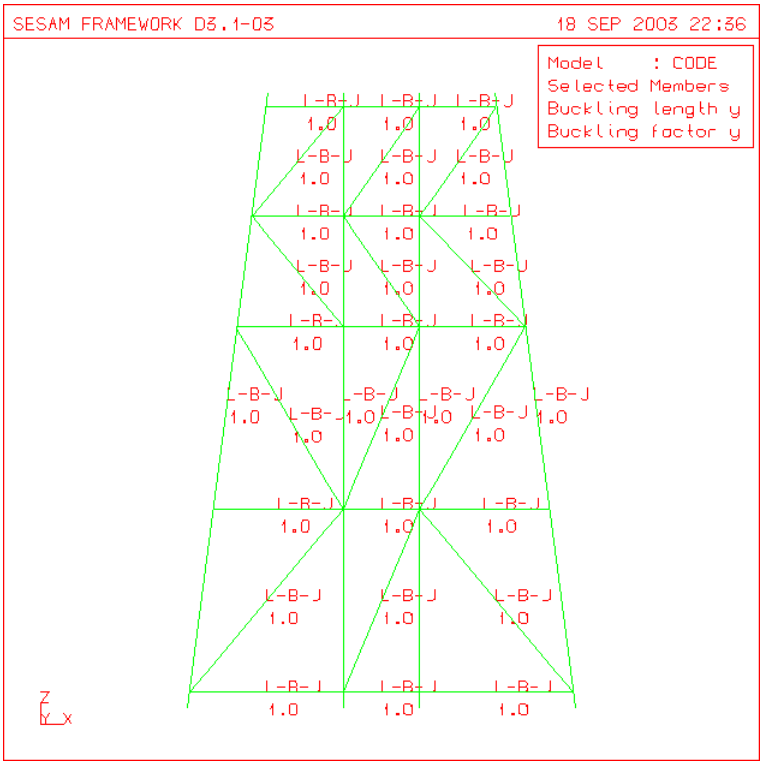

Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
20030918_220225_R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

The concept information is now read by Framework. The names of the beams close to Myjoint are shown below, and the names are identical to those in GeniE. This means that if the finite element numbering for some reason change, you can still use the same names in Framework since the beams (or concepts) are independent of mesh density.



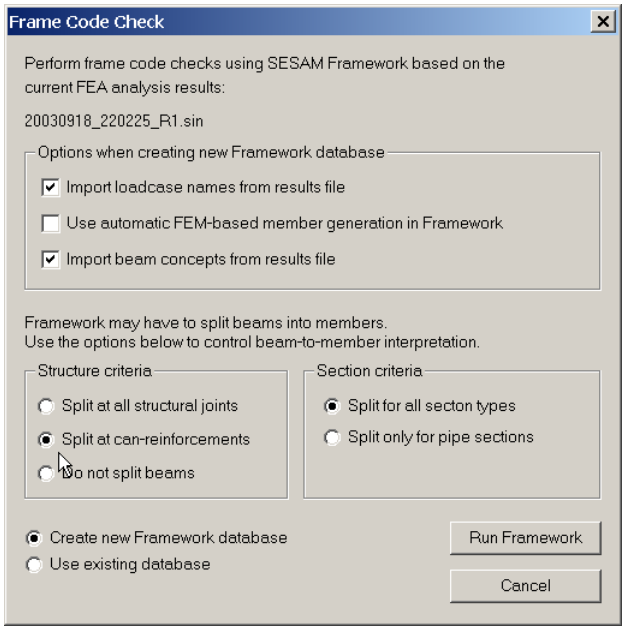
Since no split of structure is performed, the buckling length of Bm1 is identical to the member length. The pictures below show the extent of Bm1 and the buckling length of the same beam in Framework (buckling factor is 1.0, and buckling length is L-B-J (Length-Between-Joints)).





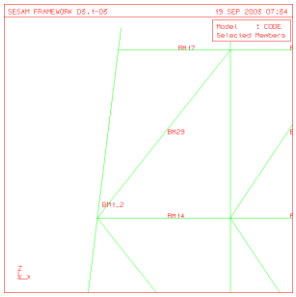
To override the buckling length factors, either specify manually or compute by using an eigenvalue approach (for tubular members only) in Framework.

11.4 Import the concept model, split structure at reinforced structural joints

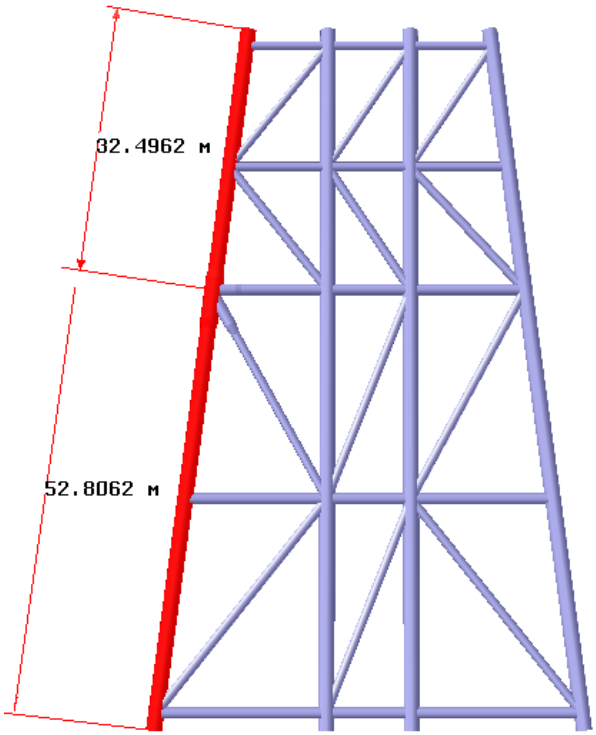

Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
20030918_220225_R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

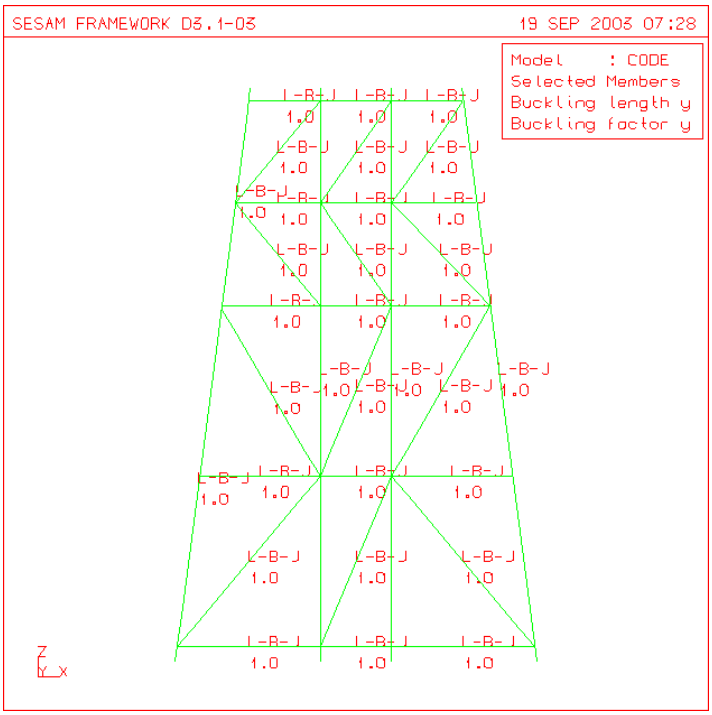
The split at can-reinforcements is the default option and will create new members in Framework. This means that you can model on a high level in GeniE and concentrate on the code checking model part of it afterwards.

The new member names will always refer to the original beam name in GeniE, for example Bm1 split in two members in Framework receive the names Bm1_1 and Bm1_2. The upper part of the leg is shown below.

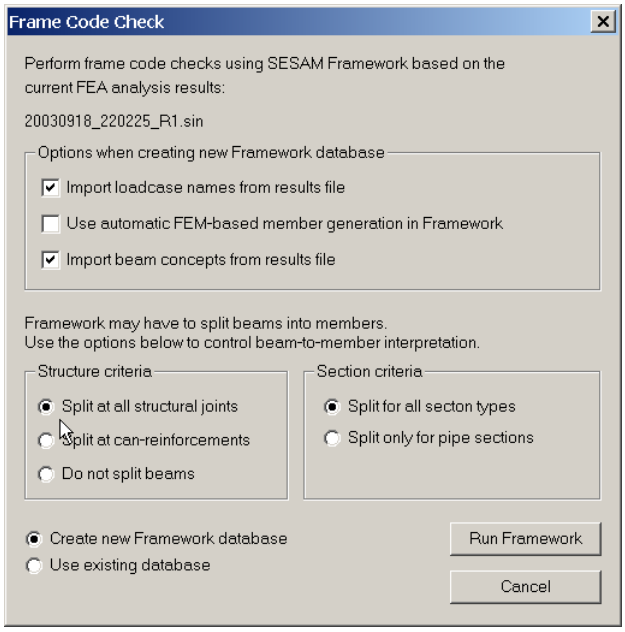


A split has been performed, and the default buckling length of the new Framework members Bm1_1 and Bm1_2 are equal to the length of the respective members. The pictures below show the extent of Bm1_1 and Bm1_2, in this case they are 52.8062 m and 32.4962 m.



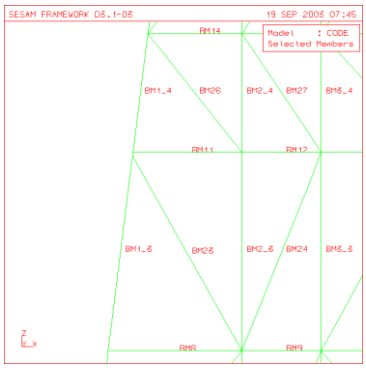


11.5 Import the concept model, split structure at all structural joints

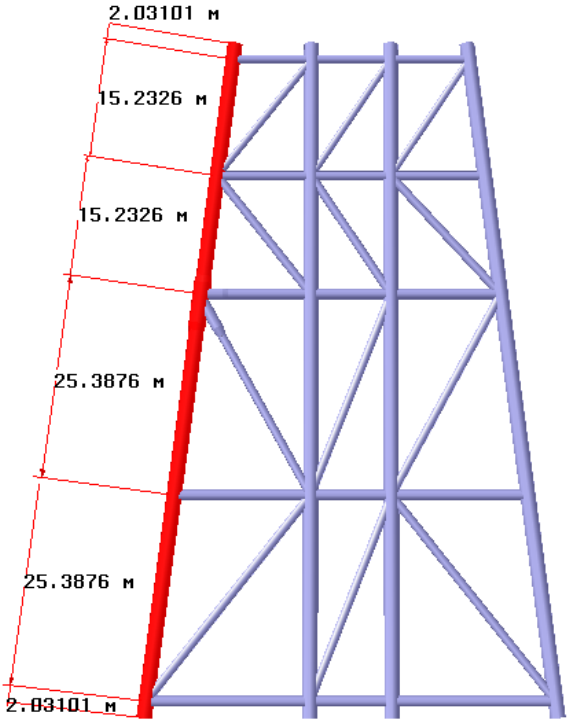

Frame Code Check
Perform frame code checks using SESAM Framework based on the current FEA analysis results:
20030918_220225_R1.sin
Options when creating new Framework database
Import loadcase names from results file
Use automatic FEM-based member generation in Framework
Import beam concepts from results file
Framework may have to split beams into members.
Use the options below to control beam-to-member interpretation.
Structure criteria
Split at all structural joints
Split at can-reinforcements
Do not split beams
Section criteria
Split for all secton types
Split only for pipe sections
Create new Framework database
Use existing database
Run Framework
Cancel

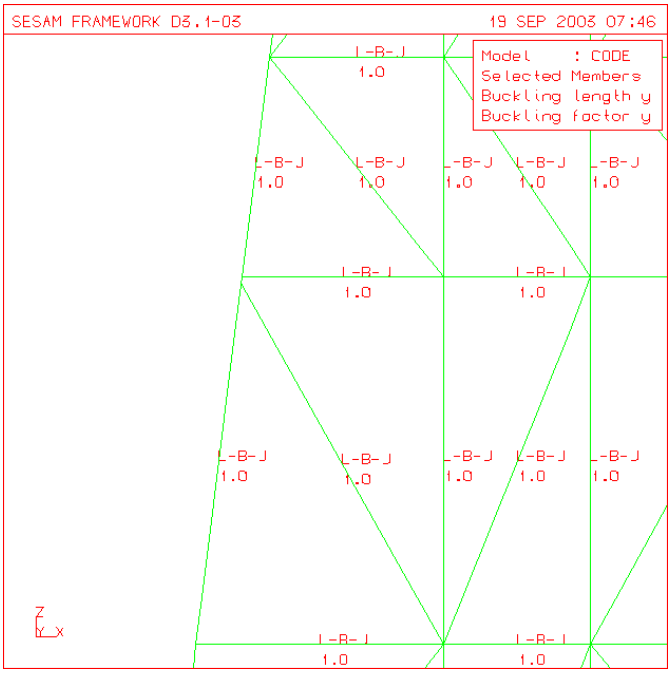
In this case the GeniE model is split at each structural joint. As such it is the same option as automatic FEM based member generation in Framework, the major difference is that concept information now is being used (beam names, sections, materials etc.).

Bm1 is now thus split in 6 members. The picture below shows the name details close to Myjoint,



The pictures below show the default buckling lengths of each member (picture focus around Myjoint).





---------- o ----------